

Rocky User Manual

Ansys
ANSYS, Inc.
Southpointe
2600 Ansys Drive
Canonsburg, PA 15317
ansysinfo@ansys.com
http://www.ansys.com
(T) 724-746-3304
(F) 724-514-9494

Release 2026 R1
March 2026

ANSYS, Inc. and ANSYS Europe, Ltd. are UL registered ISO 9001: 2015 companies.

Copyright and Trademark Information

Copyright © 2026 Synopsys, Inc. and ANSYS, Inc. This Synopsys product and all associated documentation are proprietary to ANSYS, Inc. and Synopsys, Inc. and may only be used pursuant to the terms and conditions of a written license agreement. All other use, reproduction, modification or distribution of the Synopsys product or the associated documentation is strictly prohibited.

ANSYS, Ansys Workbench, AUTODYN, CFX, FLUENT and any and all ANSYS, Inc. brand, product, service and feature names, logos and slogans are registered trademarks or trademarks of ANSYS, Inc. or its subsidiaries located in the United States or other countries. ICEM CFD is a trademark used by ANSYS, Inc. under license. CFX is a trademark of Sony Corporation in Japan. All other brand, product, service and feature names or trademarks are the property of their respective owners. FLEXlm and FLEXnet are trademarks of Flexera Software LLC.

Disclaimer Notice

THIS ANSYS SOFTWARE PRODUCT AND PROGRAM DOCUMENTATION INCLUDE TRADE SECRETS AND ARE CONFIDENTIAL AND PROPRIETARY PRODUCTS OF ANSYS, INC., ITS SUBSIDIARIES, OR LICENSORS. The software products and documentation are furnished by ANSYS, Inc., its subsidiaries, or affiliates under a software license agreement that contains provisions concerning non-disclosure, copying, length and nature of use, compliance with exporting laws, warranties, disclaimers, limitations of liability, and remedies, and other provisions. The software products and documentation may be used, disclosed, transferred, or copied only in accordance with the terms and conditions of that software license agreement.

ANSYS, Inc. and ANSYS Europe, Ltd. are UL registered ISO 9001: 2015 companies.

U.S. Government Rights

For U.S. Government users, except as specifically granted by the ANSYS, Inc. software license agreement, the use, duplication, or disclosure by the United States Government is subject to restrictions stated in the ANSYS, Inc. software license agreement and FAR 12.212 (for non-DOD licenses).

Third-Party Software

See the legal information in the product help files for the complete Legal Notice for ANSYS proprietary software and third-party software. If you are unable to access the Legal Notice, contact ANSYS, Inc.

Published in the U.S.A.

Table of Contents

1. Software Overview 31
1.1. The Ansys Product Improvement Program 32
1.2. Capabilities Charts 35
1.2.1. Rocky Particle Shapes and Capabilities 41
1.2.2. PyRocky 50
1.2.3. About This Version of Rocky 50
1.3. Program Limitations 52
1.3.1. Physics and Force Limitations 55
1.3.2. Geometry Limitations 58
1.3.3. Particle and Inlet Limitations 59
1.3.4. Materials and Interactions Limitations 62
1.3.5. SPH Limitations 63
1.3.6. Ansys and CFD Coupling Limitations 63
1.3.7. Display and Processing Limitations 66
1.3.8. Module Limitations 66
1.3.9. Known Limitations in Ansys Rocky 67
1.4. System Requirements 67
1.5. Best Practices 68
1.5.1. Best Practices for Installing a New Version of Rocky 68
1.5.2. Best Practices for Faster Processing 68
1.5.3. Best Practices for Creating Geometries for Import 70
1.5.4. Best Practices for Setting up Geometries 70
1.5.5. Best Practices for Setting Up Particles 72
2. Getting Started 73
2.1. Self-Guided Tutorials 73
2.2. User Interface 74
2.2.1. Main UI Components 74
2.2.2. Menu 75
2.2.3. Toolbar 77
2.2.4. Panels 78
2.2.5. Data Panel 79
2.2.6. Data Editors Panel 82
2.2.7. Info Tab 83
2.2.8. About the Coloring Tab 93
2.2.9. Windows Panel 97
2.2.10. Window Editors Panel 98
2.2.11. Log Panel 99
2.2.12. About the Simulation Log Panel 101
2.2.13. Cache Panel 103
2.2.14. Debug Info Panel 105
2.2.15. Undo/Redo History Panel 106
2.2.16. About Workspaces 108
2.2.17. Other UI Functionalities 112
2.3. Ansys Rocky Coupling with other Ansys Software 143
2.3.1. Ansys Rocky and Ansys Workbench 144
2.3.2. Ansys Rocky and Ansys Mechanical 170
2.3.3. Ansys Rocky and Ansys Fluent 197
2.3.4. Ansys Rocky and Ansys Motion 208
2.3.5. Ansys Rocky and Ansys optiSLang 209

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

2.3.6. Ansys Rocky and Ansys Minerva 217
2.3.7. Ansys Rocky and Ansys EnSight 217
2.4. Ansys Rocky Modules 220
2.4.1. Internal Modules (Embedded) 221
2.4.2. External Modules 222
2.4.3. Rocky Simulation Entities that can be Affected by Modules 223
2.5. Ansys Rocky Preferences 226
2.5.1. Setting Your Global Preferences 226
2.5.2. Defining Your Variables 243
2.6. Frequently Used Tasks 255
2.6.1. Enter Input Variables or Mathematical Functions as Parameter Values 256
2.6.2. Change the Speed of a Default Conveyor 258
2.6.3. Change Particle Adhesion 258
2.6.4. Set Particle Size Ranges 259
2.6.5. Use the 1-Way Fluent Method to Process Fluent and Rocky Simulations 267
2.6.6. Use the 2-Way Fluent Method to Process a Fluent and Rocky Simulation 271
2.6.7. Use the 2-Way Fluent Semi-Resolved Method to Process a Fluent and Rocky Simulation 276
2.6.8. Enable and View Particle Breakage 281
2.6.9. Create and View a Particle Assembly 283
2.6.10. Enable Thermal Modeling Calculations 284
2.6.11. Add a Logo to a 3D View 286
2.6.12. Reuse 3D View Window, Workspace, and User Process Setups in Other Projects 288
2.6.13. Set Up a New GPU Card 289
2.7. PrePost Scripting to customize Ansys Rocky software 294
2.7.1. About Creating and Using PrePost Scripts 296
2.7.2. Record a New Script 301
2.7.3. Add an External Script File to the Scripts Directory 301
2.7.4. Manually Apply a Saved Script 302
2.7.5. Automatically Apply Scripts Directly Before or After Processing 303
2.7.6. Edit a Saved Script Using the Python Programming Language 303
2.7.7. Copy a Script 304
2.7.8. Remove a Script 305
2.7.9. Rename a Script 305
2.8. Data Collection 306
2.8.1. To Enable and View Particle Breakage: 397
3. Setting Up a Simulation 411
3.1. Creating a Simulation Project 411
3.1.1. Minimum Setup Requirements 411
3.1.2. Open a Brand New Simulation Project 411
3.1.3. Open an Existing Simulation Project 412
3.1.4. Restoring an Archived Simulation Project 412
3.1.5. Begin a New Simulation Project 412
3.1.6. Open an Existing v3 Simulation Project 412
3.1.7. Open an Existing v4 or Later Simulation Project 413
3.1.8. Restore an Archived Simulation Project 413
3.2. Setting up Simulation Parameters 414
3.2.1. Study Parameters 415
3.2.2. Physics Parameters 417
3.2.3. Modules Parameters 431
3.2.4. SPH Parameters 436
3.2.5. Joints Parameters 447

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

3.2.6. Domain Settings Parameters 452
3.2.7. Solver Parameters 464
3.2.8. Geometries Parameters 480
3.2.9. Create and Edit Regions of Interest 522
3.2.10. Create and Apply Motion Frames 532
3.2.11. Create and Modify Material Compositions 591
3.2.12. Particle Sets 602
3.2.13. Point Clouds 699
3.2.14. Adding and Editing Inlets and Outlets 708
3.2.15. Collect (or Reject) Contacts Data 745
3.2.16. Set or Modify Fluid and/or Air Flow Properties 757
3.2.17. Define External Coupling Options 839
3.2.18. Duplicate a Data Panel Item 846
3.2.19. Remove Multiple Items from the Data Panel 846
3.2.20. Set Up Parameters When Restarting a Simulation 846
3.3. Saving a Simulation Project 852
3.3.1. Save a New Project 852
3.3.2. Save a Copy of a Project 852
3.3.3. Save an Archive of the Project 852
3.3.4. Save an Image of a Window 852
3.3.5. Save a New Simulation Project 853
3.3.6. Save an Exact Copy of the Simulation Project; Including the Simulation Results 853
3.3.7. Save a Copy of Only the Simulation Settings; Not the Simulation Results 854
3.3.8. Save a Copy of a Partially Processed Simulation for Restart Purposes 855
3.3.9. Save Over the Current Simulation Project 857
3.3.10. Save Custom Properties and Curves into projects 857
3.3.11. Save an Image of the Rocky Window 858
3.4. Verifying the Simulation Summary 858
3.4.1. About the Simulation Summary 859
3.4.2. Use the Simulation Summary Dialog to Verify Key Simulation Criteria 865
3.4.3. Use the Solver Summary Tab to Verify Key Simulation Criteria 866
3.4.4. Use the Solver Summary Tab to Verify the Simulation Version 866
3.5. Changing the Setup After Processing 867
3.6. Deleting the Simulation Results while keeping the setup 868
3.6.1. To delete the results of a simulation (but keep the setup): 868
3.7. Discovering Rocky Embedded Modules 868
3.7.1. Boundary Collision Statistics Module 868
3.7.2. Contacts Energy Spectra Module 870
3.7.3. Contacts Overlap Monitor Module 872
3.7.4. CFD Coupling Particle Statistics Module 873
3.7.5. Inter-group Collision Statistics Module 875
3.7.6. Inter-particle Collision Statistics Module 878
3.7.7. Intra-particle Collision Statistics Module 880
3.7.8. Multibody Dynamics FMU Coupling Module 881
3.7.9. Joint Statistics Module 882
3.7.10. About the Particle Instantaneous Energies Module 885
3.7.11. Particles Energy Spectra Module 885
3.7.12. SPH Mass Flow Rate Module 887
3.7.13. SPH Boundary Interaction Statistics Module 889
3.7.14. SPH Density Monitor Module 893
3.7.15. SPH HTC Calculator Module 895

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

3.7.16. SPH-DEM Interaction Statistics Module 899
3.7.17. SPH Surface Monitor Module 900
4. Processing a Simulation 907
4.1. Starting a Simulation 907
4.1.1. About Starting a Simulation 908
4.1.2. Start Processing a Simulation from the Beginning 913
4.1.3. Resume Processing a Stopped Simulation 913
4.1.4. Starting a Simulation with MPI (Message Passing Interface) 914
4.2. Stopping a Simulation 914
4.2.1. About Stopping a Simulation 915
4.2.2. Stop a Simulation from Processing 915
4.3. Extending the Duration of an In-Process or Completed Simulation 915
4.3.1. About Extending the Duration of an In-Process or Completed Simulation 915
4.3.2. Extend the Duration of an In-Process or Completed Simulation 919
4.4. Processing Multiple Simulations In Succession 919
4.4.1. About Processing Multiple Simulations In Succession 920
4.4.2. Open the Rocky Scheduler Program 927
4.4.3. Add a New Resource to the Rocky Scheduler 928
4.4.4. Remove a Resource from the Rocky Scheduler 928
4.4.5. Process Multiple Simulation Projects In Succession 929
5. Analyzing a Simulation 931
5.1. Use Properties and Curves 931
5.1.1. About Properties 933
5.1.2. About Curves 989
5.1.3. About Customizing Properties or Curves 1012
5.1.4. About Customizing Eulerian Statistics Properties 1017
5.1.5. About SPH Eulerian Solution 1021
5.1.6. About Adding and Editing Time Statistics Properties 1023
5.1.7. About Meshed Particles Upscaling 1033
5.1.8. Change or Remove an Existing Custom Property on the Properties or Curves Tab 1036
5.1.9. Add a New Custom Property on the Properties or Curves Tab 1037
5.1.10. Add a Property to an Eulerian Statistics User Process 1037
5.1.11. Remove a Property from an Eulerian Statistics User Process 1038
5.1.12. Add a Time Statistics Property 1039
5.1.13. Edit or Remove a Time Statistics Property 1039
5.2. Create and Modify a 3D View Window 1040
5.2.1. About 3D View Windows 1040
5.2.2. View Geometries, Particles, Points, and Fluids in 3D 1041
5.2.3. Change a 3D View Window 1042
5.2.4. Save a View with Custom Cameras 1093
5.3. View and Save Simulation Statistics 1097
5.3.1. View an Individual Statistic 1098
5.3.2. Graph (Plot or Histogram) a Data Set Within Rocky 1101
5.3.3. Export Data and Images Out of Rocky 1154
5.4. Create and Save an Animation 1166
5.4.1. About Creating and Saving an Animation 1166
5.4.2. Set up an Animation 1169
5.4.3. Use the View in the Window to Change the Qualities of a Key Frame 1171
5.4.4. Use the View in the Key Frame to Edit the Qualities of that Key Frame 1171
5.4.5. Edit the Settings and Key Frames Included in an Already Set-Up Animation 1172
5.4.6. Save an Already Set-Up Animation as an AVI 1173

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

5.4.7. Save a Static Image of Each Individual Frame in an Animation 1173
5.5. Filter Views and Data with User Processes 1174
5.5.1. Creating a User Process 1174
5.5.2. About Particles User Processes 1174
5.5.3. About Particle <-> Contact User Processes 1175
5.5.4. Analyze Data within a Defined Shape 1175
5.5.5. Analyze Data with a Clip or Cut Plane 1195
5.5.6. Analyze Data by Property Type 1202
5.5.7. Analyze the Particles Trajectory 1208
5.5.8. Inspect an Individual Cell 1211
5.5.9. Duplicate a User Process 1219
5.5.10. Export User Process Geometry Triangles to an STL File 1220
5.5.11. Analyze Particles within a Time Selection 1221
5.5.12. Divide a Cube or Cylinder User Process into Many Analysis Blocks 1226
5.5.13. Retain Particle Data Calculated within a User Process by Using Particle Calculations 1242
5.5.14. Analyze Contacts from a Sub-Set of Particles 1271
5.5.15. Analyze Particles from a Sub-Set of Contacts 1274
5.5.16. Surface User Process 1277
5.6. Change the Current Output by Using the Time Toolbar 1278
5.6.1. About the Time Toolbar 1279
5.6.2. Use the Time Toolbar to Change the Current Output 1280
5.6.3. Limit Output Values by a Specific Time Range 1281
5.6.4. Monitor Analyses Statuses by Using the Progress Panel 1281
5.6.5. About the Progress Panel 1281
5.6.6. Stop an Active Process 1283
6. Getting Help and Support 1285
6.1. Help and Support 1285
6.1.1. Ansys Engineering Copilot [BETA] 1286
6.1.2. Submit a Support Request 1293
6.1.3. Include a Simulation Project in a Support Request 1293
6.1.4. Archive a Simulation Project 1294
6.1.5. Report a Software Bug 1295
6.1.6. Verify Your Rocky License Status 1295
6.2. Remove Rocky 1296
6.2.1. To remove Rocky [Windows] 1296
6.2.2. To remove Rocky [Linux]: 1297
7. Appendices 1299
7.1. Appendix A: Particle Shape Examples 1299
7.2. Appendix B: ASCII Printable Characters 1304
7.3. Appendix F: Save Geometries as XGL, STL, or DXF Files 1307
7.3.1. Use Ansys SpaceClaim 2022 R2 to Save Geometries to STL Files 1307
7.4. Appendix G: Perform Tasks from the Command Line or Journal 1308
7.4.1. About Using Commands 1308
7.4.2. Known Limitations 1308
7.4.3. Common Commands 1309
7.4.4. Use the Command Line to Process a Simulation 1311
7.4.5. Use the Command Line to Play a Script 1313
7.4.6. Use a Journal to Run Fluent and Export Transient 1-Way Results 1313
8. Troubleshooting 1315
8.1. What kind of problems are you having? 1315
8.2. Troubleshooting Simulation Setup Problems 1315

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

8.2.1. I cannot change my import scale 1316
8.2.2. I cannot change the setup parameters 1317
8.2.3. I cannot find PrePost Scripting documentation 1317
8.2.4. I cannot find Solver SDK documentation 1318
8.2.5. I cannot get Rocky to couple with my CFD program 1319
8.2.6. I am having difficulty properly calibrating sticky, wet, or cohesive particles 1319
8.2.7. I do not know how to simulate magnetic forces 1319
8.2.8. I need help simulating an HPGR 1319
8.2.9. I cannot find the upper limit for importing custom particles shapes 1320
8.2.10. I entered an invalid variable or value and Rocky accepted it 1320
8.2.11. I briefly see long numbers when pasting in copied values 1323
8.2.12. My particle size is too large for my periodic domain 1324
8.2.13. Parts of my particle disappear or appear cut off 1324
8.2.14. Rocky does not list my Ansys version when I try to set up 2-Way Fluent Coupling 1326
8.2.15. I cannot find the Rocky component in Workbench 1327
8.2.16. I get "reference" errors in my Motion Frames when I open an older project 1327
8.2.17. I cannot find the Forces for FEM Analysis checkbox 1329
8.2.18. I get an "Unsupported Particle" error when I open an older project 1329
8.2.19. I cannot find Particle Collision Statistics 1330
8.2.20. I cannot find Particle Size Scale Factor 1330
8.2.21. I cannot find the Leeds Adhesion Force model 1331
8.2.22. I cannot find the Inlet Start and Stop Times 1331
8.2.23. I cannot find the Movements Tab for my older simulation projects 1331
8.2.24. I cannot figure out how to set the Parent Reference for a Motion Frame 1332
8.2.25. I get "Displacement" errors in my Motion Frames when I open an older project 1333
8.2.26. My "Air Viscosity" units changed 1334
8.2.27. I cannot enter an input variable or mathematical function into a text field 1334
8.2.28. I cannot find the Velocity Dependent Adhesive Force model 1335
8.2.29. I cannot find the Torque Law setting for 1-Way Constant 1336
8.2.30. I cannot find the Output Frequency parameter 1336
8.2.31. I cannot find the Loading N-Steps parameter 1336
8.2.32. I cannot find the Set Contact Detection Neighboring Distance parameter 1337
8.2.33. I cannot find the Export Solution to Rocky item in Fluent 1337
8.2.34. I cannot find the Maximum Volume Fraction Target parameter for 2-Way Fluent 1338
8.3. Troubleshooting Simulation Processing Problems 1338
8.3.1. Troubleshooting Common Processing Errors 1339
8.3.2. My simulation is taking too long to process 1351
8.3.3. I set parameters for my simulation but I cannot process it 1351
8.3.4. I get unusual results when I process my simulation 1351
8.3.5. I am processing my simulation but no output files are being created 1352
8.3.6. My particles are flowing too quickly 1353
8.3.7. Particles are going through the walls of my geometry 1353
8.3.8. No particles appear when I process the simulation 1355
8.3.9. Particles stopped flowing before my simulation was done processing 1356
8.3.10. I cannot find a batch mode for processing 1356
8.3.11. I want to use a Pascal-based GPU card instead of the recommended CUDA-based one 1356
8.3.12. I cannot find where to set the number of cores for CPU processing 1357
8.3.13. My GPU card is not being recognized as CUDA compatible 1357
8.3.14. A fiber or other particle shape disappeared from my simulation during processing 1357
8.3.15. I cannot resume processing my older simulation in the latest version of Rocky 1359
8.3.16. My inlet stopped injecting particles for a long time and then later resumed it 1359

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

8.3.17. My Volumetric Inlet isn't working right 1360
8.3.18. I cannot change my setup parameters during processing 1361
8.3.19. My particle's height increases after each collision 1362
8.4. Troubleshooting Simulation Analysis Problems 1363
8.4.1. I cannot find the zoom, pan, or tilt options on the toolbar 1364
8.4.2. I cannot see the data or statistics 1364
8.4.3. I cannot view my previous output files 1364
8.4.4. I cannot remove the colored box around my 1-Way LBM or User Process selection 1364
8.4.5. I cannot get a time counter to appear on my simulation 1365
8.4.6. I cannot get my company logo to appear on my simulation 1366
8.4.7. I cannot find the minimize, restore down, and close buttons for my maximized Rocky window 1366
8.4.8. I do not understand how power is calculated 1367
8.4.9. I want to analyze Property information for only one of my Particle sets 1368
8.4.10. I do not understand how my Particle set names map to the Particle Group value numbers .. 1368
8.4.11. I cannot find the Particle Name nor Particle Input Property 1369
8.4.12. I cannot find some Properties or Curves in this version of Rocky 1369
8.4.13. Parts of my geometries and particles disappear when I rotate my 3D View 1374
8.4.14. I cannot display or plot a particular property 1375
8.4.15. My old script no longer works 1375
8.4.16. I still want to use my old macros 1376
8.4.17. My trajectory lines are outside my periodic domain 1376
8.4.18. I have large discrepancies in my Intra-particle Collisions Statistics Frequency values 1378
8.4.19. I cannot find the directional properties for Solid Implicit Force 1379
8.5. Troubleshooting Rocky Licensing and File Problems 1379
8.5.1. Troubleshooting Common Licensing or File Errors 1379
8.5.2. Rocky says I have lost the connection with the license server 1384
8.5.3. My Rocky license will not let me open a project from the File directory 1385
8.5.4. I cannot find the import project functionality 1387
8.5.5. I cannot open my older simulation file in the latest version of Rocky 1387
8.5.6. I made a copy of my older simulation project but Rocky will not open it 1388
Glossary 1389

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information
of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

List of Figures

1.1. Particle Shape Representations 42
1.2. About Rocky screen 51
1.3. Example of correct condition for Flexible Particle Assembly 60
1.4. Example of Inappropriate condition for Flexible Particle Assembly 61
2.1. Example screenshot from Rocky Tutorial 07 - Conical Dryer (Thermal Modeling) Part A: Project Setup and Processing 73
2.2. The five main components of the Rocky user interface 75
2.3. Data Panel, default view 80
2.4. Data Panel, additional Study parameters 81
2.5. Data Editors panel for the Particles entity 82
2.6. Info tab showing results for a Feed Conveyor 83
2.7. Info tab for the Liquid Bridge Model Module 84
2.8. Info tab showing results for an individual Particle set 85
2.9. Info tab showing results for an individual Particle Input 86
2.10. Coloring tab for Particles when a 3D View window is selected 94
2.11. Coloring tab when a plot or histogram window is selected 95
2.12. Coloring tab when a Motion Frame is selected in the Data panel AND a Motion Preview window is selected 96
2.13. Coloring tab when a Particles Details window is selected 97
2.14. Windows panel showing one open 3D View window 98
2.15. The Window Editors panel showing editable options for a 3D View window 99
2.16. Simulation Log panel showing some warnings and information that was logged during processing 102
2.17. Cache Panel showing currently unused items in red 104
2.18. Cache Panel after unused items have been removed 104
2.19. Debug Info panel 106
2.20. Undo/Redo history panel showing some example actions 107
2.21. Undo/Redo history panel showing some example actions that have been undone 107
2.22. Example tabs at the bottom of the Workspace area 109
2.23. Renaming a Workspace 110
2.24. Tiling the windows on a Workspace by 3 columns 110
2.25. Example sort order in Rocky 113
2.26. Example of errors shown on the Status panel 117
2.27. Location of toggle button 120
2.28. Original plot (top); second Data panel option selected with button toggled (middle); third Data panel option selected with button toggled (bottom) 121
2.29. Edit Curves dialog for a Time Plot 122
2.30. Liquid Bridge Model Module settings showing a shared parameter 125
2.31. Directional handles for a Cube (left) and a Cylinder (right) Region of Interest 126
2.32. Directional handles for a Plane User Process 127
2.33. Example simulation log file location 136
2.34. Example simulation log file showing project information 137
2.35. Setup information shown in an example simulation log file 138
2.36. Example simulation log file showing project data per output time 139
2.37. Example simulation log file showing contacts data per output time 139
2.38. GPUs IDs shown in the simulation log file 140
2.39. GPU load balance status shown in the simulation log file 140
2.40. GPU memory status shown in the simulation log file 141
2.41. N. Halo SPH Elements and N. Enabled Coupled SPH Elements 142
2.42. Problem Partitioning Status in a DEM-only simulation 142

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

2.43. Problem Partitioning Status in a SPH-only simulation 143
2.44. Particle Dynamics (Rocky) as an Ansys Workbench component 145
2.45. Rocky Analysis System block in Workbench 145
2.46. Rocky connected to Mechanical via Workbench, showing a new External Coupling entry 147
2.47. Geometry component being modified in SpaceClaim 148
2.48. Example of a parametrized Rocky project using DesignXplorer 149
2.49. Example of static structural FEA analysis 150
2.50. Example of a transient structural FEA analysis 150
2.51. Example of a 1-Way Fluent Coupled Simulation with Rocky 151
2.52. For 1-way coupled simulations, the Initialization Method should be set as "Solver Controlled" in the Fluid Flow Solution Properties. 153
2.53. Enable export the transient data. 154
2.54. Fluent-Rocky 1-way coupling. 154
2.55. Sharing the geometry between projects. 155
2.56. 1-Way Fluent mode selected in Data Tree. 155
2.57. Data Editor for the 1-way steady-state coupling between Rocky and Fluent. 156
2.58. The coloring tab of the 1-Way Fluent Data Editor allows the selection of which fluid information should be showed in the 3D view. 157
2.59. Vectors colored by the magnitude of the fluid velocity give insight to the flow field in 1-way coupled simulations. 158
2.60. Draging a Rocky Analysis to the Project Schematics. 159
2.61. Fluent-Rocky 2-way coupling. 159
2.62. Unresolved 2-way coupling mode. 159
2.63. Enabling the Thermal in the Momentum tab of the Physics Data Editor in Rocky. 160
2.64. CFD Coupling entry in the data tree has 2-way Fluent coupling selected. 161
2.65. The Unresolved 2-way coupling mode is enabled, and the coupling setting can usually be edited. 162
2.66. The Fluent tab allows the definition of the execution mode and the data used as the initial fluid field. 163
2.67. Unresolved 2-way coupled simulation with geometry shared between Rocky and Fluent. 164
2.68. Integrated fluid and particles post-processing inside Rocky. 164
2.69. The Fluent Solution entry should be used to send the fluid data for external post-processing. 165
2.70. Input and output parameters created in Rocky are available in Workbench. 165
2.71. If something is changed in an upstream component, the Refresh option in the Setup entry should be used to update the Fluent project used in the coupled simulation. 166
2.72. The Fluent solution entry should be used for sharing the fluid data with another software, such as for structural analysis. 166
2.73. Remove Geometry component. 167
2.74. Semi-Resolved mode selected. 167
2.75. 2-Way Fluent Semi-Resolved mode selected. 168
2.76. 2-Way Fluent Semi-Resolved settings. 169
2.77. Sharing the geometry between projects. 169
2.78. Post-processing fluid data in 2-way semi-resolved coupled runs. 170
2.79. Export options for 1-way coupling with mechanical 171
2.80. Time Range Filte 172
2.81. Convert Geometry to System Coupling Wall 173
2.82. System Coupling Wall Data Panel 174
2.83. Write System Coupling File Rocky 174
2.84. System Coupling Participant file 174
2.85. Transient Thermal Ansys Mechanical 175
2.86. Details of System Coupling Region 175
2.87. System Coupling Region in Mechanical 175

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

2.88. Write System Coupling File Mechanical 176
2.89. Transient Thermal in Workbench 176
2.90. Transient Thermal in Workbench 177
2.91. Add Participant in System Coupling 177
2.92. Add Participant: System Coupling File Rocky 178
2.93. Add Participant: System Coupling File Mechanical 179
2.94. System Coupling Participants 179
2.95. Coupling Interface 180
2.96. Coupling Interface Data Panel 180
2.97. Add Data Transfer 180
2.98. Data Transfer Rocky 181
2.99. Data Transfer Rocky Setup (Temperature) 181
2.100. Data Transfer Mechanical 181
2.101. Data Transfer Mechanical Setup (Heat Flow) 182
2.102. Data Transfer Rocky and Mechanical 183
2.103. Solution Control 184
2.104. Output Control 185
2.105. Start Solve 185
2.106. System Coupling After Solving 186
2.107. Access EnSight through System Coupling 186
2.108. Post Processing EnSight 186
2.109. Changing the Translation 187
2.110. Dragging the desired parameter to the Geometry 188
2.111. Geometries Solved 188
2.112. Parameter Solved added to the geometry 189
2.113. Coupling Types for the System Coupling Wall 190
2.114. Write System Coupling File Rocky 191
2.115. Transient Structural Ansys Mechanical 191
2.116. Project data tree for the Transient Strctural 192
2.117. Time Step in Analysis Settings for the Transient Structural 193
2.118. Write System Coupling Files 194
2.119. Add FSI Data Transfers inside System Coupling 195
2.120. Force and Incremental Displacement Data Transfers 195
2.121. Ansys Ensight Post-Processing 196
2.122. Ansys Rocky Post-Processing 196
2.123. Thermal coupled simulation schematics. 198
2.124. 2-way Fluent Boundaries tab in the Data Editors 199
2.125. Select the walls in the Load Fluent Boundaries. 200
2.126. New entries under Geometries in the Data panel. 200
2.127. Options available for a coupled wall. 201
2.128. Options available for a standard wall. 201
2.129. Additional transient properties for coupled walls. 202
2.130. The wall temperature, initially at 300K, increases as hot particles (500K) collide with the wall. 203
2.131. Select Wall Thermal Zones in the Rocky Export plugin. 203
2.132. List of all the walls compatible with the 1-way thermal coupling between Rocky and Fluent in the Thermal Wall Selector window. 204
2.133. Walls selected in the Thermal Wall Selector window. 204
2.134. Results exported to Rocky using the Rocky Export Plugin. 205
2.135. Fluent (Fluid -> Particle) Selected. 205
2.136. 1-way Fluent Boundaries tab in the Data Editors. 205
2.137. Load Fluent Boundaries listing all the compatible boundaries. 206

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

2.138. Walls to be selected for the thermal coupling. 206
2.139. New entries under Geometries in the Data panel. 207
2.140. Options available in Coupled Walls. 207
2.141. Properties created for the coupled walls. 208
2.142. New Project 210
2.143. Solver Wizard 210
2.144. **Ansys Rocky selection in Ansys optiSLang software** 211
2.145. **Project file Selection** 212
2.146. **Choose the Rocky script to export results** 212
2.147. **Enabling Beta Modules in Ansys optiSLang** 213
2.148. **Ansys Rocky (Beta) selection in Ansys optiSLang software** 214
2.149. **Ansys Rocky (Beta) selection in Modules Dock** 214
2.150. **Ansys Rocky (Beta) Module Solver Settings** 215
2.151. **Ansys Rocky (Beta) Module Settings Tab** 216
2.152. **Ansys Rocky (Beta) Module PrePost Scripts** 216
2.153. **Ansys Rocky (Beta) Module Schematic view** 217
2.154. Tutorial 01 inside Ansys Rocky 218
2.155. Opening Ansys EnSight 218
2.156. Selecting the .rocky file 219
2.157. Tutorial 01 inside Ansys EnSight 219
2.158. Tutorial 01 render by Ansys EnSight 220
2.159. Modules available by default in Rocky 222
2.160. Additional Features 227
2.161. Preferences, 3d View 228
2.162. Preferences, Multi Time Plot (similar for all plots) 230
2.163. Preferences, Histogram 231
2.164. Preferences, Cross Plot Axis (similar for all graphs) 235
2.165. Preferences, Multi Time Plot Layout 237
2.166. Example of entering a variable into a text field 245
2.167. The variable name and units used for the field appears after double-clicking the field 245
2.168. The current value for the variable entered appears in the field after clicking away from the field 245
2.169. Edit Properties dialog for Output variables 252
2.170. Comparison of the three different Size Types in Rocky 260
2.171. Example Size Distribution Settings for Sieve Size 261
2.172. Surface wear modification 262
2.173. Color map of surface intensity 263
2.174. Surface wear modification 264
2.175. Color map of surface intensity 266
2.176. Example of Instantaneous Breakage being simulated in an HPGR machine 281
2.177. 3D View window with a Sample Logo added as an Image Overlay 287
2.178. Error seen in Rocky when TDR causes the GPU to stop processing 290
2.179. Python Shell panel 295
2.180. PrePost Script panel showing different scripts on both tabs 297
2.181. Solver, PrePost Scripts sub-entity in Data Editors panel 298
2.182. PrePost Script panel showing no scripts shared across projects 299
2.183. New PrePost Script dialog 299
2.184. Example Contacts overlap message on the Simulation Log panel 309
2.185. Options in the Data Editors panel when the Boundary Collision Statistics Module is enabled 313
2.186. Boundary Collision Properties available for geometries after processing 314
2.187. Boundary Collision Curves available for geometries after processing 315
2.188. Color map of surface intensity 316

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

2.189. Options in the Data Editors panel when the Inter-group Collision Statistics Module is enabled 319
2.190. Additional module options for Particle groups when the Inter-group Collision Statistics module is enabled 320
2.191. Additional module options for a geometry component when the Inter-group Collision Statistics module is enabled 321
2.192. Curves for Particles (simulation-wide) when Inter-group Collision Statistics are enabled 322
2.193. Options in the Data Editors panel when the Intra-particle Collision Statistics Module is enabled 325
2.194. Rocky error shown if no particle sets in the simulation support Intra-particle Collisions Statistics 326
2.195. Particles Details window showing Stress Normal collision statistics for a particle set 327
2.196. Particle | Modules sub-tab available when Enable per Group Statistics is selected 331
2.197. Options in the Data Editors panel when the Inter-particle Collision Statistics Module is enabled 332
2.198. Properties available for the main Particles entity when Inter-particle Collision Statistics is enabled 333
2.199. Options in the Data Editors panel when the CFD Coupling Particle Statistics Module is enabled 337
2.200. Properties available for the main Particles entity when CFD Coupling Particle Statistics is enabled 338
2.201. There are no options in the Data Editors panel when the Particle Instantaneous Energies Module is enabled 342
2.202. Properties for Particles (simulation-wide) when Particle Instantaneous Energies is enabled 343
2.203. Curves for Particles (simulation-wide) when Particle Instantaneous Energies is enabled 344
2.204. Energy : Potential property for Particles shown in a 3D View window 345
2.205. Options in the Data Editors panel when the Contacts Energy Spectra Module is enabled 351
2.206. Additional module options for Particle groups when the Contacts Energy Spectra module is enabled 352
2.207. Additional module options for a geometry component when the Contacts Energy Spectra module is enabled 352
2.208. SAG Mill Simulated with both Grinding Media (Steel Balls) and Ore (Rock) 353
2.209. Examples of Contacts Impact Energy Spectra curves for various particle-to-particle and particle-to-geometry contact pairs 354
2.210. Curves tab for Particles showing three sections for Energy 355
2.211. Example cross plot of Contacts Energy Spectra showing both Cumulative Power and Power, for impact collisions between particles and geometries 356
2.212. Options in the Data Editors panel when the Particles Energy Spectra Module is enabled 359
2.213. Additional module options for Particle groups when the Particles Energy Spectra module is enabled 360
2.214. SAG Mill Simulated with both Grinding Media (Steel Balls) and Ore (Rock) 361
2.215. Particles Energy Spectra curves related to Impact Energy 362
2.216. Particles Energy Spectra comparison 363
2.217. Curves tab for Particles showing three sections for Specific Energy 364
2.218. Example cross plot showing Specific Shear Energy Curves 365
2.219. Example cross plot showing energy spectra values 367
2.220. Example cross plot showing energy spectra values 369
2.221. Comparison of the three different Size Types in Rocky 376
2.222. Example Size Distribution Settings for Sieve Size 377
2.223. Surface wear modification 378
2.224. Color map of surface intensity 379
2.225. Surface wear modification 380
2.226. Color map of surface intensity 382
2.227. Example of Instantaneous Breakage being simulated in an HPGR machine 397
2.228. 3D View window with a Sample Logo added as an Image Overlay 402
2.229. Error seen in Rocky when TDR causes the GPU to stop processing 405
3.1. Study parameters in the Data Editors panel (default, unsaved project) 415
3.2. Study parameters in the Data Editors panel (saved project with information entered) 416

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

3.3. Physics, Gravity parameters in the Data Editors panel 419
3.4. Physics, Momentum parameters in the Data Editors panel 419
3.5. Physics, Thermal parameters in the Data Editors panel 420
3.6. Physics, Coarse-Graining parameters in the Data Editors panel 420
3.7. Modules available by default in Rocky 432
3.8. Info tab for the Liquid Bridge Model Module 435
3.9. SPH Model Parameters 438
3.10. SPH Model Advanced Parameters 443
3.11. Joints parameters in the Data Editors panel (Coloring - Joints) 448
3.12. Joints parameters in the Data Editors panel (Coloring - Joints Connectivity) 449
3.13. Joints parameters in the Data Editors panel (default, unsaved project) 451
3.14. Joints properties in the Data Editors after simulation (default, unsaved project) 452
3.15. Particle behavior when exiting the coordinate limits 453
3.16. Particle behavior when exiting only the coordinate limits (left) and then with a periodic cartesian domain set (right) 454
3.17. Geometry triangles lying fully on the periodic plane (in green) should be removed or disabled 455
3.18. Comparison between internal surface and Full 3D geometry imports 456
3.19. Illustration of how the size and location of a Cylindrical Periodic Domain is determined 457
3.20. Cylindrical Periodic Domain with full wedge (left) and with center removed (right) 458
3.21. Periodic planes intersecting boundary triangles (left) and aligning on the edges (right) 459
3.22. Domain Settings, Coordinate Limits parameters in the Data Editors panel (No periodic domain) 460
3.23. Domain Settings, Cartesian Periodic Domain Type parameters 460
3.24. Domain Settings, Cylindrical Periodic Domain Type parameters 460
3.25. Solver | Time (default) parameters in the Data Editors Panel 465
3.26. Solver | Time parameters for 2-Way Fluent CFD Coupling simulations in the Data Editors panel 466
3.27. Solver | General parameters in the Data Editors panel 469
3.28. Solver | Advanced tab (Advanced Features) settings in the Data Editors panel 470
3.29. Solver | Output Properties 478
3.30. Geometry options available in Rocky. 481
3.31. Convert Wall into Surface. 482
3.32. Convert Surface into Wall. 482
3.33. Options available when creating a rectangular or circular surface 484
3.34. Import File Info dialog 488
3.35. Options available for geometries imported as surfaces 490
3.36. Example of a mill liner being modified by wear 493
3.37. Replication example showing how one bucket in a bucket elevator can be turned into 20 494
3.38. Imported wall, Geometry parameters in the Data Editors panel 495
3.39. Imported wall, Mass parameters in the Data Editors panel 496
3.40. Imported wall, Wear parameters in the Data Editors panel 497
3.41. Imported wall, Replication parameters in the Data Editors panel 498
3.42. **Imported wall, Thermal parameters in the Data Editors panel** 499
3.43. **Imported wall, SPH parameters in the Data Editors panel** 500
3.44. Imported wall, Modules tab in the Data Editors panel 500
3.45. Default Feed Conveyor, Geometry parameters in the Data Editors panel 509
3.46. Default Feed Conveyor, Orientation parameters in the Data Editors panel 509
3.47. Default Feed Conveyor, Skirtboard parameters in the Data Editors panel 510
3.48. Default Feed Conveyor, Feeder Box parameters in the Data Editors panel 510
3.49. Default Feed Conveyor, Head Pulley parameters in the Data Editors panel 510
3.50. Default Feed Conveyor, Belt Profile parameters in the Data Editors panel 511
3.51. Default Feed Conveyor, Belt Motion parameters in the Data Editors panel 511
3.52. Default Feed Conveyor, Modules tab in the Data Editors panel 511

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

3.53. Default Receiving Conveyor, Geometry parameters in the Data Editors panel 515
3.54. Default Receiving Conveyor, Orientation parameters in the Data Editors panel 515
3.55. Default Receiving Conveyor, Skirtboard parameters in the Data Editors panel 516
3.56. Default Receiving Conveyor, Belt Profile parameters in the Data Editors panel 516
3.57. Default Receiving Conveyor, Belt Motion parameters in the Data Editors panel 516
3.58. Default Receiving Conveyor, Modules tab in the Data Editors panel 516
3.59. Select Output Unit dialog 520
3.60. Comparison of a User Process Shape with a Region of Interest Shape 523
3.61. Cube Region of Interest during project setup 524
3.62. Cylinder arc results with different Final angle values 527
3.63. Cylinder Region of Interest 528
3.64. Cylinder shape examples: arch (left), slice (center), and tube (right) 528
3.65. How to create a Region of Interest 531
3.66. Sine wave for a vibration when the Initial Phase is zero (default value) 534
3.67. Sine wave for a vibration when the Initial Phase is 90 degrees 535
3.68. Frame <01> with two child Frames before duplication 539
3.69. After Frame <01> is duplicated, Frame <02> appears with two child Frames 539
3.70. Motion Frames parameters in the Data Editors panel, and the Motion Preview window 541
3.71. Parameters for newly added Frame entry - Angle and Vector 542
3.72. Parameters for newly added Frame entry - Basis Vectors 543
3.73. Parameters for newly added Frame entry - Angles 544
3.74. Motion Frame with Enable Periodic Motion selected 545
3.75. New Motion added to Frame entry showing Translation parameters 545
3.76. New Motion showing Rotation parameters 546
3.77. New Motion showing Periodic Rotation (Pendulum) parameters 546
3.78. New Motion showing Periodic Translation (Vibration) parameters 546
3.79. New Motion showing Free Body Translation parameters 547
3.80. New Motion showing Free Body Rotation parameters 547
3.81. New Motion showing Additional Force parameters 547
3.82. New Motion showing Additional Moment parameters 547
3.83. New Motion showing Spring-Dashpot Force parameters 548
3.84. New Motion showing Spring-Dashpot Moment parameters 548
3.85. New Motion showing Linear Time Variable Force parameters 548
3.86. New Motion showing Linear Time Variable Moment parameters 548
3.87. New Motion showing Time Series Translation parameters 549
3.88. New Motion showing Time Series Rotation parameters 550
3.89. Parameters for newly added Cone Crusher Frame 566
3.90. Components of a Motion Preview Window 572
3.91. Motion Preview View tab options on the Window Editors panel 576
3.92. Coloring tab when a Motion Frame is selected in the Data panel AND a Motion Preview window is selected 580
3.93. Example Coloring tab in the Data Editors panel showing a geometry component 581
3.94. Materials parameters in the Data Editors panel 592
3.95. Default Material parameters in the Data Editors panel 593
3.96. Fluid materials parameters in the Data Editors panel 596
3.97. Materials Interactions parameters in the Data Editors panel for a Constant Adhesive Force 598
3.98. Main Particle tab on the Data Editors panel 603
3.99. Particle, Shape tab on the Data Editors panel showing a Particles Details window 606
3.100. Particle, Shape tab on the Data Editors panel for a default particle shape 607
3.101. Particle, Shape tab on the Data Editors panel for an Assembly particle shape 608
3.102. Particle, Shape tab after importing a Custom Polyhedron shape 609

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

3.103. Particle, Shape tab on the data Editors panel after a Custom Shell is imported 609
3.104. Particle, Size tab on the Data Editors panel 610
3.105. Comparison of the three different Size Types in Rocky 611
3.106. Particle, Size tab on the Data Editors panel when Coarse Grain Modeling is enabled 612
3.107. Particle, Orientation tab on the Data Editors panel 613
3.108. A default-orientated Particle set being released from an inlet 614
3.109. A Particle set with a 45 degree Rotation Angle along the X axis being released from an inlet 615
3.110. A Particle set that allows random orientations within 45 degrees of the X axis being released from an inlet 616
3.111. Particles Details window showing example particle rotated 90 degrees in the Z direction 617
3.112. Particle, Movement tab on the Data Editors panel 618
3.113. Particle, Composition tab on the Data Editors panel 621
3.114. Particle, Composition tab on the Data Editors panel for Shell particles 622
3.115. Particle, Composition tab for Fibers when both Anisotropic and Bilinear Elastoplastic Joint Model are enabled 623
3.116. Example error messages shown when a Target Number of Elements value is below the minimum calculated by Rocky 625
3.117. Original shape compared with its meshed version with different Feature Angle settings 626
3.118. Particle, Breakage tab on the Data Editors panel 627
3.119. Particle, Breakage tab on the Data Editors panel showing Ab-T10 model settings 628
3.120. Particle, Breakage tab on the Data Editors panel showing the Tavares model settings 629
3.121. Particle, Breakage tab on the Data Editors panel showing the Griffith Energy Criterion model settings 630
3.122. Particle, Breakage tab on the Data Editors panel showing the Shear Stress Criterion model settings ... 630
3.123. Particle, Breakage tab on the Data Editors panel showing the Tensile Stress Criterion model settings 631
3.124. Particle, Breakage tab on the Data Editors panel showing the Tensile or Shear Stress Criterion model settings 631
3.125. Particle, Breakage tab on the Data Editors panel showing the von Mises Stress Criterion model settings 632
3.126. Particle, Breakage, Fragments sub-tab on the Data Editors panel showing the Gaudin-Schumann distribution model 633
3.127. Particle, Breakage, Fragments sub-tab on the Data Editors panel showing the Incomplete Beta distribution model 634
3.128. Particles, Info tab on the Data Editors panel 635
3.129. Particles, Deformation tab (Plastic) on the Data Editors panel 636
3.130. Particles, Modules tab on the Data Editors panel 637
3.131.Traditional (imported) concave bolt shape (left) compared with the same shape defined as an Assembly particle (right) 664
3.132. Example of Instantaneous Breakage in Rocky 665
3.133. Example of Discrete Breakage in Rocky 666
3.134. Illustration showing the effect of weakening due to accrual of damage in repeated loading events 668
3.135. Convert to Convex message you might see after importing a Custom Polyhedron shape 671
3.136. Example Custom Fiber defined in a text file 672
3.137. Example four-Segmented fiber and the .csv file used to define it 674
3.138. Example .csv file used with modules that models the custom joint behavior 674
3.139. Import File Info dialog 676
3.140. Particles Details window overview when used to preview a particle shape 680
3.141. Particles Details window overview when used to view collision statistics after processing 680
3.142. Particles Details window and Window Editors panel settings 683
3.143. Particles Details View tab options on the Window Editors panel 684

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

3.144. Initial Overlays tab options on the Window Editors panel showing default Particle Set text 685
3.145. Image overlay options on the Window Editors panel 686
3.146. Example Coloring tab in the Data Editors panel showing Particle Set parameters for a Particles Details window 696
3.147. Select Output Unit dialog 698
3.148. Example text file defining a Static Point Cloud with two variables and two points 700
3.149. Example text file defining the main file of a Transient Point Cloud with three Static Point Clouds. 701
3.150. Example text file defining the main file of a Transient Point Cloud with three Static Point Clouds. 701
3.151. Example text files structure to define a Transient Static Point Cloud with three Static Point Clouds. 702
3.152. A Point Cloud being visualized in a 3D View window 703
3.153. Point Cloud parameters in the Data Editors panel 704
3.154. How to create a Point Cloud 705
3.155. How to set the Time Periodic for the Point Cloud 706
3.156. How to set the Motion Frame for the Point Cloud 707
3.157. Search Cut -Off options for the Point Cloud 707
3.158. Example of Particle Inlet from a Circular Surface 709
3.159. Example of a Particle Volumetric Inlet 710
3.160. Example of a Volumetric Inlet being constrained by a drum geometry (wall or surface) 710
3.161. Example of a Volumetric Inlet following along the walls of the pipe geometry to which the inlet was constrained 711
3.162. Comparison of different Gap Scale Factors in a Volumetric Inlet 711
3.163. Comparison between spherical and non-spherical particles in a Volume-fill 713
3.164. Different methods of limiting the Volumetric Inlet 714
3.165. Broom example simulated with Particle Custom Inlets and Frozen Fibers 718
3.166. Particle Inlet, Particles sub-tab, Data Editors panel 720
3.167. Particle Inlet, Time sub-tab, Data Editors panel 720
3.168. Particle Inlet, Entry sub-tab, Data Editors panel 721
3.169. Fluid Inlet, SPH sub-tab, Data Editors panel 726
3.170. Fluid Inlet, Time sub-tab, Data Editors panel 726
3.171. Volumetric Inlet, Particles sub-tab, Data Editors panel 729
3.172. Volumetric Inlet, SPH sub-tab, Data Editors panel 729
3.173. Volumetric Inlet, Region sub-tab, Data Editors panel 730
3.174. Volumetric Inlet, Time sub-tab, Data Editors panel 731
3.175. Particle Custom Inlet, Data Editors panel 736
3.176. Outlet, Particles tab 738
3.177. Outlets, SPH tab 739
3.178. Frictional contacts data being visualized with particles (top) and without particles (bottom) 747
3.179. Contacts Properties options on the Data Editors panel 749
3.180. Particle-to-particle contact data visualized as line segments being shown with particles (top) and without particles (bottom) 750
3.181. Contacts parameters on the Data Editors panel (Coloring - Contacts) 752
3.182. Contacts parameters on the Data Editors panel (Coloring - Contacts Network) 753
3.183. Contacts parameters on the Data Editors panel 755
3.184. Dynamic placement of CFD coupling 758
3.185. 1-Way Constant method showing shapes of different densities falling through water 759
3.186. 1-Way LBM method showing how dust-like vectors interact with the particles 760
3.187. 1-Way Fluent with steady state fluid data showing how particles interact with the fluid-like vectors ... 761
3.188. 2-Way Fluent method showing colored particles reacting to a fluidized bed 762
3.189. CFD Coupling, 1-Way Constant in the Data Editors panel 764
3.190. CFD Coupling, 1-Way Constant, Fluid Flow Tab in the Data Editors panel 765
3.191. CFD Coupling, 1-Way LBM tab in the Data Editors panel 772

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

3.192. Rocky Export menu in Fluent 776
3.193. Example of a (steady state) 1-Way Fluent simulation showing static fluid flow vectors (in blue), and transient particle nodes + vectors (in red) 780
3.194. The Additional Variables Selector dialog in Fluent (left). These become custom Properties in Rocky after F2R import (right). 781
3.195. The Fluent to Rocky (F2R) time-step multiplier dialog in Fluent 781
3.196. Properties panel for Fluent Solution 791
3.197. The Ansys Fluent Coupling Support option selected during Rocky installation 793
3.198. Rocky install screen showing Ansys Workbench Addin selected 794
3.199. Illustration showing Fluent species information (left, red box) being shared with Rocky (right, red box) during 2-Way Fluent setup. Later during processing, Rocky sends mass source terms (right, blue box) back to Fluent for all but the last species in the list. 799
3.200. Volumetric Diffusion Method 802
3.201. Mapping Across Interfaces 803
3.202. Example of a 2-Way Fluent simulation showing transient fluid flow vectors (in purple), and transient particle nodes (all colors) 804
3.203. Lift and Torque Laws in the Data Editors panel 805
3.204. Lift and Torque Laws in the Data Editors panel after processing your simulation 806
3.205. CFD Coupling, 2-Way Fluent, Interactions tab in the Data Editors panel 807
3.206. CFD Coupling, 2-Way Fluent, Coupling tab in the Data Editors panel 811
3.207. CFD Coupling, 2-Way Fluent, Coupling tab when Uniform Distribution is selected for Mapping Method 811
3.208. 2-Way Fluent, Zones and Interfaces tab 816
3.209. CFD Coupling, 2-Way Fluent, Fluent tab in the Data Editors panel 817
3.210. 2-Way Fluent, Fluent tab when Local Parallel is selected for Execution mode 817
3.211. 2-Way Fluent, Fluent tab when Distributed Parallel is selected for Execution mode 818
3.212. CFD Coupling, 2-Way Fluent, Variables tab in the Data Editors panel 822
3.213. 2-way Coupled Rocky-Fluent simulation in Workbench initializing the fluid field using the values set in the Fluent setup 824
3.214. Fluent tab in Rocky Data Editors when coupling within workbench without initial fluid data file 825
3.215. 2-way Coupled Rocky-Fluent simulation in Workbench initializing the fluid field using an initial fluid field data 826
3.216. Fluent tab in Rocky Data Editors with existing fluid data is used in a 2-way coupled simulation 827
3.217. CFD Coupling, 2-Way Fluent Semi-Resolved in the Data Editors panel 834
3.218. Workbench best practices for Rocky-Mechanical coupling. 840
3.219. External Coupling entity expanded to show the Geometry Loads component 841
3.220. Physics thermal tab when HTC transfer between Rocky and Mechanical via Workbench is used. 843
3.221. SPH HTC Calculator module enabled. 844
3.222. Physics thermal tab when HTC transfer between Rocky and Mechanical via Workbench is used. 844
3.223. Save Custom Properties In the Project or Locally 857
3.224. Custom Properties saved In the Project and Locally 858
3.225. Simulation Summary dialog 860
3.226. Solver, Summary tab on the Data Editors panel before processing 861
3.227. Solver, Summary tab on the Data Editors panel during or after processing 862
3.228. Options in the Data Editors panel when the Boundary Collision Statistics Module is enabled 869
3.229. Options in the Data Editors panel when the Contacts Energy Spectra Module is enabled 870
3.230. Additional module options for Particle groups when the Contacts Energy Spectra module is enabled 871
3.231. Additional module options for a geometry component when the Contacts Energy Spectra module is enabled 871
3.232. Example Contacts overlap message on the Simulation Log panel 872

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

3.233. Options in the Data Editors panel when the Contacts Overlap Monitor Module is enabled 872
3.234. Options in the Data Editors panel when the CFD Coupling Particle Statistics Module is enabled 874
3.235. Options in the Data Editors panel when the Inter-group Collision Statistics Module is enabled 876
3.236. Additional module options for Particle groups when the Inter-group Collision Statistics module is enabled 877
3.237. Additional module options for a geometry component when the Inter-group Collision Statistics module is enabled 878
3.238. Options in the Data Editors panel when the Inter-particle Collision Statistics Module is enabled 879
3.239. Options in the Data Editors panel when the Intra-particle Collision Statistics Module is enabled 880
3.240. Options in the Data Editors panel when the Multibody Dynamics FMU Coupling is enabled 881
3.241. Options in the Data Editors panel when the Joint Statistics Module is enabled 883
3.242. Statistic data in joint properties 884
3.243. There are no options in the Data Editors panel when the Particle Instantaneous Energies Module is enabled 885
3.244. Options in the Data Editors panel when the Particles Energy Spectra Module is enabled 886
3.245. Additional module options for Particle groups when the Particles Energy Spectra module is enabled 887
3.246. Options in the Data Editors panel when the SPH Mass Flow Rate Module is enabled 888
3.247. Geometries options in the Data Editors panel when the SPH Mass Flow Rate Module is enabled 889
3.248. Options in the Data Editors panel when the SPH Boundary Interaction Statistics Module is enabled ... 890
3.249. Results in Walls Properties when the SPH Boundary Interaction Statistics Module is enabled 892
3.250. Results in Walls Curves when the SPH Boundary Interaction Statistics Module is enabled 893
3.251. Options in the Data Editors panel when the SPH Density Monitor Module is enabled 894
3.252. Simulation Log example when the SPH Density Monitor Module is enabled 894
3.253. Example application of heat transfer estimation using the SPH HTC Calculator module. 895
3.254. SPH HTC Calculator module options. 896
3.255. New wall properties are shown in the Data Editors panel when the SPH HTC Calculator module is enabled. 898
3.256. Options in the Data Editors panel when the SPH DEM Interaction Statistics Module is enabled 899
3.257. Options in the Data Editors panel when the SPH Surface Monitor Module is enabled 901
4.1. Extend Simulation dialog 917
4.2. Rocky Scheduler with one case running and two in the queue (single-instance license) 921
4.3. Rocky Scheduler with 2 cases running simultaneously and one in the queue (unnumbered license) 922
4.4. Add Resource dialog for CPU Resource Type 923
4.5. Add Resource dialog for a single GPU Resource Type 923
4.6. Add Resource dialog for multiple GPUs Resource Type 923
4.7. Assign Resources dialog 924
5.1. Example Properties tab showing all available data columns 935
5.2. Time Scope option on the Properties tab for all entities 936
5.3. Standard Properties options on the Data Editors panel for Geometry components 937
5.4. Collisions Statistics Properties on the Data Editors panel for Geometry components 938
5.5. Particles, Standard Property options in the Data Editors panel 946
5.6. Particles, Property options in the Data Editors panel when Inter-particle Collision Statistics are collected 947
5.7. Particles, Property options in the Data Editors panel when Particle Instantaneous Energies are collected 948
5.8. Particles, Property options in the Data Editors panel when CFD Coupling Particle Statistics are collected 949
5.9. Supported Particle set, Properties options on the Data Editors panel when Intra-particle Collision Statistics are collected 965
5.10. Contacts Properties options on the Data Editors panel 970

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

5.11. SPH Properties options on the Data Editors panel 973
5.12. 1-Way LBM, Properties options on the Data Editors panel 976
5.13. Fluent 1-Way Instantaneous, Properties options in the Data Editors panel 978
5.14. Fluent 1-Way variables are selected during the export process 979
5.15. 2-Way Fluent Properties in the Data Editors panel 981
5.16. Eulerian Statistics (applied to a Cube), Properties options in the Data Editors panel 984
5.17. Conventional index notation for stress tensors within a Cube-shaped selection 986
5.18. Diagram showing how Motion Frames assigned to a rectangular boundary affect the resulting velocity Curves for the boundary 991
5.19. Geometry (Feed; Receiving; Imported) Curves options on the Data Editors panel when Boundary Collision Statistics are collected 992
5.20. Imported Geometry, Curves options on the Data Editors panel when Motion Frames are assigned 992
5.21. Particles, Curves options on the Data Editors panel 995
5.22. Particles, Curves options on the Data Editors panel when Inter-group Collision Statistics is enabled 997
5.23. Particles, Curves options when Particle Instantaneous Energies is collected 1002
5.24. Particles, Curves options when Energy Spectra is collected 1004
5.25. Solver, Curves options on the Data Editors panel 1006
5.26. Motion Frames, Curves options on the Data Editors panel 1010
5.27. Add/Edit custom (property or curve) button locations 1013
5.28. Add new dialog (property or curve) 1013
5.29. Custom Property dialog for Simple Expressions 1015
5.30. Custom Curves dialog for Advanced Expressions 1015
5.31. Add and edit properties for source entity button location 1018
5.32. Add or edit eulerian properties dialog 1018
5.33. Add new property dialog 1018
5.34. Properties available in the SPH Eulerian Solution 1022
5.35. A rotating mill at a single time step 1024
5.36. The same mill and time step as above, but showing one Property (standard Property) 1025
5.37. The same mill and time step as Figure 2 but with Eulerian Statistics applied showing the average values of a Property within each Eulerian block (custom Property) 1026
5.38. The same mill and Eulerian Statistics applied as above, but now those values are averaged over a set range of time (Time Statistics Property) 1027
5.39. Add and edit time statistics properties button location 1028
5.40. Edit time statistics properties dialog showing two new Properties 1028
5.41. Add time statistics properties dialog 1029
5.42. Two newly added Time Analysis Properties in the Data Editors panel 1030
5.43. Example results when meshed particle upscaling is enabled (left) and disabled (right) 1033
5.44. Meshed Particles Upscaling, Study parameter 1034
5.45. 3D View window showing particle flow after processing 1041
5.46. Mouse with center scroll wheel 1043
5.47. Camera Visualization toolbar 1045
5.48. Colored Legend options that can be changed 1049
5.49. Window Axes options that can be changed 1050
5.50. Particle rendering scale options 1051
5.51. Grid-functions options on the Window Editors panel when the main 3D View entity is selected 1052
5.52. Color-scale dialog 1054
5.53. Calculate limits for time range dialog 1055
5.54. Axes options on the Window Editors panel when the main 3D View entity is selected 1056
5.55. Particles options on the Window Editors panel when the main 3D View entity is selected 1057
5.56. Examples of text and image overlays in a 3D View window 1059
5.57. 3D View options on the Window Editors panel 1060

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

5.58. Initial Overlay options for 3D View windows on the Window Editors panel 1061
5.59. Text Overlay options on the Window Editors panel 1061
5.60. Image Overlay options on the Window Editors panel 1062
5.61. Window Axes options that can be changed 1072
5.62. Axes options on the Window Editors panel when the main 3D View entity is selected 1072
5.63. A Cube User Process with Eulerian Statistics applied displays Number of Particles with Show on Nodes disabled (left) and enabled (right) 1075
5.64. A custom conveyor showing Solid Color Faces with Edges showing Intensity : Impact 1076
5.65. A custom conveyor showing 70% Transparent Solid Color Faces with increased Point Size Nodes showing Intensity : Impact 1077
5.66. A custom conveyor with Faces showing Intensity : Impact with Show on Nodes disabled (left) and enabled (right) 1077
5.67. Example of a 2-Way Fluent simulation showing transient fluid flow vectors (in purple), and transient particle nodes (all colors) 1078
5.68. Example Coloring tab in the Data Editors panel showing 1-Way LBM 1079
5.69. Coloring tab in the Data Editors panel for a Point Cloud 1080
5.70. Coloring toolbar 1084
5.71. Show Annotations provides a default text overlay for geometries 1085
5.72. Color Scale options that can be changed on the Coloring tab 1087
5.73. Coloring tab options for Color Scale entry in the Data Editors panel 1088
5.74. Color-scale dialog 1090
5.75. Calculate limits for time range dialog 1091
5.76. Custom Presets toolbar 1094
5.77. Custom Presets toolbar 1095
5.78. Create new preset dialog 1095
5.79. Property column header options in the Data Editors panel 1099
5.80. Example Time Plot 1103
5.81. Inspecting a Time Plot Curve with Shift+Click (top left), Ctrl+Click (top right), and Ctrl+Click+Drag (bottom) 1104
5.82. Table tab results of the example Time Plot 1106
5.83. Add Curve Expression dialog 1107
5.84. Edit Expression dialog 1108
5.85. Inspecting a Multi-Time Plot Curve with Shift+Click (top left), Ctrl+Click (top right), and Ctrl+Click+Drag (bottom) 1110
5.86. Example Multi Time Plot showing three sub-plots 1111
5.87. Example Histogram 1112
5.88. Example Histogram showing a comparison of weighted statistics 1113
5.89. Example Histogram showing a cumulative percentage of weighted statistics 1114
5.90. Location of the Configure histogram button 1115
5.91. Configure Histogram dialog showing one statistic weighted by another 1116
5.92. Example of a Particle Trajectory property being graphed in a Space Plot 1119
5.93. Example Cross Plot 1120
5.94. Inspecting a Cross Plot Dot-Cloud with Shift+Click (top left), Ctrl+Click (top right), and Ctrl+Click+Drag (bottom) 1121
5.95. Simulated particle pile to be measured 1122
5.96. Cross Plot of the above particle pile showing the Particle X-Coordinate position crossed with the Particle Y-Coordinate position 1123
5.97. Window Editors panel tabs for a Multi Time Plot 1127
5.98. Time Plot settings in the Window Editors panel 1128
5.99. Time Plot, Curves display in the Window Editors panel 1132
5.100. Time Plot, Axes settings in the Window Editors panel 1133

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

5.101. Time Plot, Annotations | Label settings in the Window Editors panel 1136
5.102. Time Plot, Annotations | Values settings in the Window Editors panel 1137
5.103. Time Plot, Annotations | Symbol settings in the Window Editors panel 1138
5.104. Time Plot, Annotations | Text settings in the Window Editors panel 1139
5.105. Time Plot, Annotations | Position settings in the Window Editors panel 1140
5.106. Time Plot, Annotations | Border settings in the Window Editors panel 1141
5.107. Multi Time Plot, Layout settings in the Window Editors panel 1144
5.108. Example Annotation added to a Time Plot 1146
5.109. Location of the Annotations menu on the graph window 1146
5.110. Create Annotation dialog 1147
5.111. Annotation Created 1148
5.112. Coloring tab when a plot or histogram window is selected for the Particles entity 1153
5.113. A Time Plot window in Rocky (left); the saved snapshot image of the same window (right) 1156
5.114. Export Image settings in the Window Editors panel (Plots and Histograms only) 1156
5.115. Export settings in the Window Editors panel (3D Views, Motion Preview, and Particles Details windows only) 1157
5.116. Time Plot, Export Curves settings in the Window Editors panel 1158
5.117. Export Configuration dialog 1158
5.118. Example of force exported CSV file 1160
5.119. Example of pressure exported CSV file 1161
5.120. Export Loads dialog 1162
5.121. Animation panel 1167
5.122. How to create a Cube Group (in the image: from Particles and SPH) 1177
5.123. Example of Cube Group (each Cube inherits Motion Frame, Center, Magnitude and Orientation from their group - Cube Group <01>, in the example). 1178
5.124. Cube User Process applied to the Particles entity 1179
5.125. How to create a Cylinder Group (in the image: from Particles and SPH) 1182
5.126. Example of Cylinder Group (each Cylinder inherits Motion Frame, Center, Magnitude and Orientation from their group - Cylinder Group <01>, in the example). 1183
5.127. Cylinder arc results with different Final angle values 1184
5.128. Cylinder User Process applied to the Particles entity 1185
5.129. Cylinder shape examples: arch (left), slice (center), and tube (right) 1185
5.130. Import File Info dialog 1187
5.131. Polyhedron (Envelope) User Process showing an imported almond shape 1188
5.132. Surface User Process Properties 1189
5.133. Surface User Process with 3D tool for manipulation 1190
5.134. Streamline User Process applied 1193
5.135. Clip Plane (left) and Cut Plane (right) examples as applied to a Feed Conveyor. Note that because Select Mode was chosen for the Cut Plane (right side of figure above), the two triangles making up the return conveyor are also included in the selection. 1196
5.136. How to create a Plane Group (in the image: from Particles and SPH) 1197
5.137. Example of Plane Group (each Plane inherits Motion Frame, Center, Magnitude and Orientation from their group - Plane Group <01>, in the example). 1198
5.138. Plane User Process applied to the Particles entity 1199
5.139. Plane User Process applied to a geometry component 1199
5.140. Filter User Process with Select Mode applied to a Eulerian Statistics User Process - Data Editors Panel 1203
5.141. Filter User Process with Select Mode applied to a Eulerian Statistics User Process - Particles and Eulerian Statistics (On/Off) 1204
5.142. Filter User Process with Cut Mode applied to a Eulerian Statistics User Process - Data Editors Panel ... 1204

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

5.143. Filter User Process with Cut Mode applied to a Eulerian Statistics User Process - Particles and Eulerian Statistics (On/Off) 1205
5.144. Filter User Process applied to a custom conveyor - Data Editors Panel 1206
5.145. Filter User Process applied to a custom conveyor 1206
5.146. Particle flow (top) compared with Particles Trajectory User Process (bottom). 1209
5.147. Inspector tab on the Data Editors panel for a Geometry Cell Inspector User Process (left); Geometry triangle selected for inspection (right) 1212
5.148. Inspector tab on the Data Editors panel for a Particles Cell Inspector User Process (left); individual whole particle selected for inspection (right) 1213
5.149. Inspector tab on the Data Editors panel for a Particles Cell Inspector User Process (left); individual particle fragment selected for inspection (right) 1213
5.150. Inspector tab on the Data Editors panel for a Eulerian Statistics Cell Inspector User Process (left); Eulerian Statistics block selected for inspection (right) 1214
5.151. Inspector tab on the Data Editors panel for a 1-Way LBM Air Flow Cell Inspector User Process (left); Air Flow cell selected for inspection (right) 1215
5.152. Inspector tab on the Data Editors panel for a 1-Way Fluent Inspector User Process (left); 1-Way Fluent node selected for inspection (right) 1216
5.153. Inspector panel showing a Geometry Cell Inspector User Process 1217
5.154. Select Output Unit dialog 1220
5.155. Example screening device designed to sort particles based upon their size 1222
5.156. Example histogram showing the PSD of the particles that exited at the bottom (red) compared to the PSD of the particles that entered the domain (orange) 1223
5.157. Particles Time Selection User Process applied to the Particles entity 1224
5.158. A Cube User Process applied to Particles with a Eulerian Statistics User Process applied to the Cube with a Properties User Process applied to the Eulerian Statistics, which results in only the blocks with particles being displayed in the 3D View 1228
5.159. A rotating mill slice with particles 1229
5.160. The same mill slice as above but with Eulerian Statistics applied showing Volume Fraction 1230
5.161. The same mill slice with Eulerian Statistics but with a Property User Process applied limiting the blocks shown to only those with a Volume Fraction higher than 0.25 1231
5.162. The same mill with Eulerian Statistics blocks limited by Volume Fraction but now with the blocks colored by Absolute Translational Velocity 1232
5.163. A Slurry Mill with DEM Particles and SPH Elements 1233
5.164. Eulerian User Process created from a Cylinder User Process 1233
5.165. Adding Properties to the Eulerian User Process 1234
5.166. Properties tab showcasing the Custom Properties created 1235
5.167. Eulerian User Process colored by DEM Mass Fraction 1236
5.168. Eulerian User Process colored by SPH Mass Fraction 1237
5.169. Cube User Process from the Particles entity 1238
5.170. Cube User Process from the Particles entity with Eulerian Statistics applied 1238
5.171. Cube User Process showing red (local X), green (local Y) and blue (local Z) directional handles 1239
5.172. Cylinder User Process from the Particles entity 1239
5.173. Cylinder User Process from the Particles entity with Eulerian Statistics applied 1240
5.174. Cylinder User Process with Eulerian Statistics displaying 3 radial, 10 tangential, and 2 axial divisions . 1240
5.175. Residence Time applied to Particles and a Cube User Process shape respectively (top); Particles Calculations for Residence Time shown for Particles and the Cube User Process shape respectively (bottom). 1243
5.176. Residence Time Calculation 1244
5.177. Flip Count applied to Particles and a Cube User Process shape respectively (top); Particles Calculations for Flip Count shown for Particles and the Cube User Process shape respectively (bottom). 1245
5.178. Mill slice illustration showing how a single particle moving through a Cube User Process is calculated for Flip Count Particles Calculations. 1246

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

5.179. Particles Calculations for Residence Time shown on the Properties tab for Particles 1247
5.180. Particles Calculations for Residence Time shown on the Properties tab for the Cube User Process 1248
5.181. Mixing equipment at the beginning of the simulation with tagged particles in the center 1249
5.182. Mixing equipment later in the simulation with tagged particles mixed 1250
5.183. Histogram showing Absolute Translational Velocity of the tagged particles 1251
5.184. Mixing equipment with Divisions Tagging applied to create 5 separately tagged groups 1252
5.185. Mixing equipment later in the simulation with tagged groups of particles beginning to mix 1253
5.186. Mixing equipment at the end of the simulation with a Cube User Process selection applied to the center portion 1254
5.187. Histogram comparing a center sample of mixed particles to the initial composition 1255
5.188. Particles with Divisions Tagging showing a Tag Stride of 2 1256
5.189. Particles with Divisions Tagging after being mixed for 20 seconds 1257
5.190. Particle Tagging option on the Data panel inside Calculations 1258
5.191. Particle Tagging option on the Data panel inside Particles 1259
5.192. Particle Tagging on the Data Editors panel 1260
5.193. Particle Calculations, Divisions Tagging options (for a Cube) on the Data Editors panel 1261
5.194. SPH Tagging option on the Data panel inside Calculations 1265
5.195. SPH Tagging option on the Data panel inside SPH 1266
5.196. SPH Tagging on the Data Editors panel 1267
5.197. Property process created from Particles (top view) showing desired particle group in yellow; Particle to Contact process created from Property process showing Normal Forces (bottom view) 1272
5.198. Cube from Contacts (in green, top view); Contact to Particle from Cube colored by Absolute Rotational Velocity (bottom view) 1275
5.199. Surface User Process Properties 1278
5.200. Time toolbar showing the output time at 0.06s (3rd Output file) 1279
5.201. Time Toolbar with slider highlighted in yellow to indicate its use during Motion Preview 1279
5.202. Time Filter dialog 1280
5.203. Progress panel showing the status of a Time plot request 1282
6.1. License Management dialog showing a License Type of Ansys 1296
8.1. PrePost Scripting Manual 1317
8.2. List of application-level PrePost Scripting functionality available for use in your PrePost Scripts 1318
8.3. Warning provided in Status panel when Troughing Angle value is above the recommended maximum 1321
8.4. Warning provided in Status panel when Troughing Angle is below the recommended minimum 1322
8.5. Error shown on Data Editors panel when zero is entered for Troughing Angle 1322
8.6. Example of a long number (1) truncated (2), copied (3), and then pasted outside of Rocky (4) 1323
8.7. Simulation setup failed message 1324
8.8. Example of a small-scale straight fiber (above) and a small-scale sphero-polyhedron (below) displaying rendering and lighting issues in the Particles Details window 1325
8.9. Rocky install screen showing Ansys Workbench Addin selected 1327
8.10. Two different Motion Frame references errors appearing in the Data panel 1328
8.11. Example of a particle set with an Unsupported Particle error 1329
8.12. Example showing how nested Frames can recreate a "parent" reference 1332
8.13. A Motion Frame displacement error appearing in the Data panel 1333
8.14. Example error when Rocky attempts to access a regular project file with a long path 1343
8.15. Example error when Rocky attempts to access a restored archive project file with a long path 1344
8.16. Simulation setup failed message for long file paths 1345
8.17. Simulation setup failed message for too many particles 1346
8.18. Example "surface velocity changed" warning on the Simulation Log panel 1346
8.19. Example "contacts overlap" message on the Simulation Log panel 1347
8.20. Simulator critical message reporting "Error copying memory" 1348

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

8.21. Simulation setup failed error 1349
8.22. Example showing simulated Sphere particles going through a geometry wall 1353
8.23. Example showing simulated particles "bleeding" through a geometry wall 1354
8.24. A fiber particle just before the center of one of its elements reaches the domain limit and disappears from the simulation 1358
8.25. Example overlap warning found on the Simulation Log panel 1359
8.26. Simulation setup failed error 1360
8.27. Simulation time keyword added to a 3D View as a text overlay 1365
8.28. Sample logo added to a 3D View as an image overlay 1366
8.29. Button locations for a maximized window in Rocky 1367
8.30. Feed Conveyor with parts that disappear from the 3D View when rotated 1374
8.31. Incorrect trajectory lines when using Cylindrical Periodic Domains 1377
8.32. Cube User Processes added over the Particles Trajectory serves to filter out the incorrect lines 1377
8.33. Mesh A is treated by Rocky as a convex shape; Mesh B is treated as a concave shape 1378
8.34. Simulator message 1380
8.35. Example Links removed message 1381
8.36. Example message you might see if you try opening a project that has been locked 1383
8.37. Example message you might see if you try saving a project that has been locked 1383
8.38. License error announcing lost connection 1385
8.39. Example - License Management dialog showing a Status of No license for feature 1386

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information
of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

List of Tables

1.1. Particle Shape Type and Capabilities Matrix 42
1.2. Flexible Particle Capabilities (Rocky Professional and Rocky Academic Only) 46
2.1. CFD Coupling Particle Statistics Collection Options 339
3.1. Joints Parameters - Coloring 449
3.2. Particle Custom Inlet Definition File Field Definitions 715
3.3. Fluid Inlet parameter options 726
3.4. Contacts Parameters - Coloring 753
5.1. Surface User Process Parameter Options 1190
5.2. Streamlines parameter options in Data Editors 1193

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information
of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

Chapter 1: Software Overview

Rocky is a 3D Discrete Element Method (DEM) modeling program that simulates how a set of particles (p. 1393) interact with other particles and the various boundaries (p. 1389) that represent the conveyor chute, mill, mixer, plow, coater or other kind of bulk materials handling equipment being analyzed.

Engineers use Rocky to test and adjust geometries (p. 1391) based upon the metrics Rocky provides. In this way, Rocky helps manufacturing organizations:

- Increase equipment life and capacity
- Eliminate blockages, and belt and liner punctures
- Reduce power draw, dust, and noise
- Decrease spillage and product degradation
- Enhance mixing; minimize dead zones and segregation
- Evaluate scale-up strategies
- Predict force, torque, and power consumption
- Analyze structural loads by coupling with FEA
- Simulate fluid-granular interactions by coupling with CFD

Access more Rocky resources at AIS Rocky

To learn more about DEM, refer to the following resources:

- [Video] What is DEM?
- [Webinar] What is DEM and its Application in the Industry?

Rocky What's New

Each new release of Rocky includes a list of what features and functionalities have changed since the last released version. This list is displayed automatically whenever you choose to upgrade your version. Or, to review the list at any other time, from the **Help** menu, click **Release Notes**.

This section includes the following topics:

- The Ansys Product Improvement Program (p. 32)
- Package and Version Capabilities (p. 35)
- Program Limitations (p. 52)

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

- System Requirements (p. 67)
- Best Practices (p. 68)

1.1. The Ansys Product Improvement Program

This product is covered by the Ansys Product Improvement Program, which enables Ansys, Inc., to collect and analyze *anonymous* usage data reported by our software without affecting your work or product performance. Analyzing product usage data helps us to understand customer usage trends and patterns, interests, and quality or performance issues. The data enable us to develop or enhance product features that better address your needs.

How the Program Works

The Ansys Product Improvement Program, controlled through the Licensing Settings utility, is enabled by default. During each session, the product collects anonymous usage data. When the session ends, this data is securely transmitted to a server accessible only to authorized Ansys employees. After Ansys receives the data, statistical methods, such as distributions, counts, means, medians, and modes, are applied to analyze and understand usage patterns.

Data We Collect

The data we collect under the Ansys Product Improvement Program are limited. The types and amounts of collected data vary from product to product. Typically, the data fall into the categories listed here:

Hardware: Information about the hardware on which the product is running, such as the:

- brand and type of CPU
- number of processors available
- amount of memory available
- brand and type of graphics card

System: Configuration information about the system the product is running on, such as the:

- operating system and version
- country code
- time zone
- language used
- values of environment variables used by the product

Session: Characteristics of the session, such as the:

- interactive or batch setting
- time duration

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

- total CPU time used
- product license and license settings being used
- product version and build identifiers
- command line options used
- number of processors used
- amount of memory used
- errors and warnings issued

Session Actions: Counts of certain user actions during a session, such as the number of:

- project saves
- restarts
- meshing, solving, postprocessing, etc., actions
- times the Help system is used
- times wizards are used
- toolbar selections

Model: Statistics of the model used in the simulation, such as the:

- number and types of entities used, such as nodes, elements, cells, surfaces, primitives, etc.
- number of material types, loading types, boundary conditions, species, etc.
- number and types of coordinate systems used
- system of units used
- dimensionality (1-D, 2-D, 3-D)

Analysis: Characteristics of the analysis, such as the:

- physics types used
- linear and nonlinear behaviors
- time and frequency domains (static, steady-state, transient, modal, harmonic, etc.)
- analysis options used

Solution: Characteristics of the solution performed, including:

- the choice of solvers and solver options
- the solution controls used, such as convergence criteria, precision settings, and tuning options

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

- solver statistics such as the number of equations, number of load steps, number of design points, etc.

Specialty: Special options or features used, such as:

- user-provided plug-ins and routines
- coupling of analyses with other Ansys products

Data We Do Not Collect

The Product Improvement Program does *not* collect any information that can identify you personally, your company, or your intellectual property. This includes, but is not limited to:

- names, addresses, or usernames
- file names, part names, or other user-supplied labels
- geometry- or design-specific inputs, such as coordinate values or locations, thicknesses, or other dimensional values
- actual values of material properties, loadings, or any other real-valued user-supplied data

In addition to collecting only anonymous data, we make no record of where we collect data from. We therefore cannot associate collected data with any specific customer, company, or location.

Opting Out of the Program

You may *stop* your participation in the program any time you wish. See the Enabling the Ansys Product Improvement Program (APIP) documentation in the Ansys, Inc. Licensing Guide for more information.

The Ansys, Inc., Privacy Policy

All Ansys products are covered by the Ansys, Inc., Privacy Policy.

Frequently Asked Questions

1. *Am I required to participate in this program?*

   No, your participation is voluntary. We encourage you to participate, however, as it helps us create products that will better meet your future needs.

2. *Am I automatically enrolled in this program?*

   Yes. All users are automatically enrolled in the program.

3. *Does participating in this program put my intellectual property at risk of being collected or discovered by Ansys?*

   No. We do not collect any project-specific, company-specific, or model-specific information.

4. *Can I stop participating even after I agree to participate?*

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

Yes, you can stop participating at any time. See the Enabling the Ansys Product Improvement Program (APIP) documentation in the Ansys, Inc. Licensing Guide for more information.

5. *Will participation in the program slow the performance of the product?*

   No, the data collection does not affect the product performance in any significant way. The amount of data collected is very small.

6. *How frequently is data collected and sent to Ansys servers?*

   The data is collected during each use session of the product. The collected data is sent to a secure server once per session, when you exit the product.

7. *Is this program available in all Ansys products?*

   Not at this time, although we are adding it to more of our products at each release. The program is available in a product only if this *Ansys Product Improvement Program* description appears in the product documentation, as it does here for this product.

8. *If I enroll in the program for this product, am I automatically enrolled in the program for the other Ansys products I use on the same machine?*

   Yes. Your enrollment choice applies to all Ansys products you use on the same machine. Similarly, if you end your enrollment in the program for one product, you end your enrollment for all Ansys products on that machine.

9. *How is enrollment in the Product Improvement Program determined if I use Ansys products in a cluster?*

   In a cluster configuration, the Product Improvement Program enrollment is determined by the host machine setting.

10. *Can I easily opt out of the Product Improvement Program for all clients in my network installation?*

    Yes. See the Enabling the Ansys Product Improvement Program (APIP) documentation in the Ansys, Inc. Licensing Guide for more information.

1.2. Capabilities Charts

The features and functionalities that Rocky has are listed in the capability charts provided below:

Note:

The capabilities listed here are for the base Rocky program. Additional capabilities are available via ready-to-use external modules (p. 1391). Refer to the Install an External Module (p. 223) topic for details.

Table 1: General Features
Solid Particles DEM-Discrete Element Method
Fluid SPH-Smoothed Particles Hydrodynamic
Modify Material Compositions (p. 591)

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

Table 1: General Features
Modify Materials Interactions (p. 597)
Extend Simulation (p. 915)
Restart Capabilities (p. 855)
Resume Capabilities (p. 913)
Solver Scheduler Tool (p. 920)
Periodic Domains (Cartesian or Cylindrical) (p. 452)
PrePost Scripting Functionality (for Creating PrePost Scripts) (p. 296)
Solver SDK (Software Development Kit) (for Creating Custom External Modules (p. 431)) **Note:** Separate SDK download required
Parametric Support (Pre & Post) (p. 372)
Unit System Management (p. 238)
Project Archive & Restore (p. 1294)
Windows OS Support
Linux OS Support
Command Line & Headless System Support (p. 1308)
Light and Dark Themes
PyRocky (p. 50)

Table 2: Additional Features
SPH Density Monitor Module
Advanced Features (p. 227)
Experimental (Beta) Features (p. 227)
External Modules (p. 223) including Ansys Motion (p. 208) Coupling, Liquid Bridge Model, TBM, and more.
Regions of Interest (ROIs) (p. 522) (for use with external modules (p. 1391) only)
Point Clouds (p. 699) (for use with external modules (p. 1391) only)
Material Wizard
Calibration Suite
Solver Modules
Ansys Motion FMU through Ansys Mechanical Export

Table 3: Parallel Processes
Parallel Shared Memory CPU Solver (p. 908) (HPC licenses required)

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

Table 3: Parallel Processes
SingleGPU and MultiGPU Solver (p. 908) (HPC licenses required)

Table 4: Contact and Physical Models
Rolling Resistance Model (p. 417): Type A: Constant Moment (Type 1)
Rolling Resistance Model (p. 417): Type C: Linear Spring Rolling Limit (Type 3)
Contact Model (p. 417) (Normal Forces): Hysteretic Linear Spring, Linear Spring Dashpot, and Hertzian Spring Dashpot
Contact Model (p. 417) (Tangential Forces): Linear Spring Coulomb Limit, Coulomb Limit, and Mindlin-Deresiewicz
Contact Model (p. 417) (Adhesive Forces): Constant, Linear, and JKR
Shear Wear Contour Plots (p. 381)
Wear (Surface Modification) (p. 379)
Instantaneous Breakage Models: Ab-T10 and Tavares (p. 665)
Discrete Breakage Models (p. 665): Griffith Energy Criterion, Shear Stress Criterion, Tensile Stress Criterion, Tensile or Shear Stress Criterion, and von Mises Stress Criterion
Thermal Model (p. 400)
CFD Coupling: 1-Way LBM (p. 771)
CFD Coupling: 1-Way Constant (p. 763)
Coarse Graining (p. 417)
Joint Models (p. 602): Linear Elastic and Bilinear Elastoplastic
Laminar and Turbulent Fluid Models
Surface Tension Models

Table 5: Geometry Types
Walls Surfaces (Used for Inlets, Outlets and Flow Measurement)
Rocky Feed Conveyor Parametric
Rocky Receiving Conveyor Parametric
Imported Geometries (Walls or Surfaces) (p. 487) from Ansys Fluent (import *.cas and *.msh Files)
Imported Geometries (Walls or Surfaces) (p. 487) from Ansys Motion (p. 208) (import *.fmu and *.dfg Files)

Table 6: Geometry Motions
Rotation Motion (p. 533)
Translation Motion (p. 533)
Rotation Motion (p. 533) Without Displacement
Translation Motion (p. 533) Without Displacement

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

Table 6: Geometry Motions
Periodic Translation (Vibration) Motion (p. 533)
Periodic Rotation (Pendulum) Motion (p. 533)
Additional Force/Moment Motion (p. 533)
Spring-Dashpot Force/Moment Motion (p. 533)
Linear Time Variable Force/Moment Motion (p. 533)
Cone Crusher Motion (p. 567)
Periodic Motion (p. 533)
6-DoF (Degrees of Freedom) Motion (p. 533)
Imported Motions from Ansys Fluent (p. 533)
Replicate Geometry
Nested Motion (p. 533) Frames
2-Way Coupling with Ansys Motion (p. 208)

Table 7: Particle Types
Spherical (p. 1395) Particle Type (p. 602) (Sphere)
Shaped (p. 1395) Particle Types (p. 602) including Polyhedron, Briquette, Faceted Cylinder, Sphero-Cylinder, Sphero-Polygon, and Sphero-Polyhedron
Custom Imported (p. 670) Convex Particles
Custom Imported (p. 670) Concave Particles
Flexible Particle Type (p. 602) (Fiber, Shell, and Polyhedron)
Fiber Particle Type (p. 602) (Rigid, Flexible, Frozen, Discrete Breakage, and/or CFD Coupled)
Shell Particle Type (p. 602) (Rigid, Flexible, Discrete Breakage, and/or CFD Coupled)
Assembly Particle Type (p. 602) (Rigid, CFD Coupled and *Flexible as Beta feature*)
Particle Size Distribution (p. 375)
Adhesive Materials (p. 374) (Wet/Dry/Sticky Contact Rheologies)

Table 8: Inlets and Outlet Types
Particles and Fluid Inlets
Volumetric Inlet (Particles and Fluid)
Custom Particle Inlet
Particles and Fluids Outlets

Table 9: Export Options
Image Export (p. 858)
Property and Curves Table Data Export (p. 1164)
Geometry Export (p. 520), including Modified Wear Boundaries

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

Table 9: Export Options
Particle Shape Export (p. 698)
Project Context Export (p. 403)

Table 10: Data Gathering and Quantitative Post-Processing
Cube Selection (p. 1176)
Cylinder Selection (p. 1181)
Plane Selection (p. 1195)
Polyhedron Envelope Selection (p. 1186)
Property Filter (p. 1202)
Particle Residence Time Property (p. 933)
Particle Tagging (p. 1248)
Eulerian Statistics (p. 1226)
Particles Time Selection (p. 1221)
Particles Trajectory (p. 1208)
Particle to Contact (p. 1271) and Contact to Particle (p. 1274) Selections
Particle Flip Count Property (p. 933)
Limit Calculations to Particle Selection (p. 1242)
Default Properties and Curves (p. 931)
Time Statistics Properties (p. 1023)
Custom Properties and Curves (p. 1012)
Intra-particle Collision Statistics (p. 324)
Particles Energy Spectra (p. 358)
Contacts Energy Spectra (p. 350)
Boundary Collision Statistics (p. 312)
Inter-particle Collision Statistics (p. 331)
Contacts Analysis (p. 746)
Inter-group Collision Statistics (p. 319)
Particle Instantaneous Energies (p. 341)
Joint Statistics
SPH Mass Flow Rate
SPH Boundary Interaction Statistics
Heat Transfer Coefficient (HTC) calculator
SPH-DEM Interaction Statistics
Streamlines User Process
Surface User Process for SPH Flow measurement

Table 11: Visualization Tools
3D View (p. 1040)

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

Table 11: Visualization Tools
Time Plot (p. 1102)
Space Plot (p. 1118)
Histogram (p. 1112)
Cross Plot (p. 1120)
Video Animation (p. 1166)
Motion Preview (p. 572)
Multi-Time Plot (p. 1110)
Particle and Cell Inspector (p. 1211)
Particles Details 3D View (p. 679)
Custom Preset Camera Views (p. 1094)

Table 12: Ansys Coupling
Parametric Integration through Ansys Workbench (p. 144)
2-Way Thermal Coupling
Ansys Mechanical Coupling
1-Way Static Structural Coupling (DEM-to-FEA (p. 1165))
1-Way Transient Structural Coupling (DEM-to-FEA (p. 1165))
1-Way Transient Thermal Coupling (DEM-to-FEA (p. 1165))
1-Way Static Thermal Coupling (DEM-to-FEA (p. 1165))
2-Way Thermal Coupling (DEM-FEA (p. 170))
2-Way Structural Coupling (DEM-FEA (p. 170))
Ansys Fluent Coupling
Static 1-Way Fluent Coupling (CFD-to-DEM) (p. 775)
Transient 1-Way Fluent Coupling (CFD-to-DEM) (p. 775)
Single Phase 2-Way Coupling (CFD-DEM) (p. 795)
Multiphase 2-Way Coupling (CFD-DEM) (p. 795)
Multi-Species 2-Way Coupling (CFD-DEM) (p. 795)
2-Way Fluent Semi-Resolved Coupling (p. 830)
Boundary Thermal Coupling for 2-Way Fluent Unresolved Coupling
Other Integrations with Ansys
2-Way Coupling with Ansys Motion (p. 208)
Ansys Minerva Integration (p. 217)
Ansys optiSLang Integration (p. 209)
Ansys EnSight Integration

ANSYS ROCKY STUDENT SOFTWARE

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

Ansys is committed to setting today's students up for success tomorrow, by providing free simulation engineering software licenses to students at all levels.

Ansys Rocky software is available in a student version and can be downloaded in the link below. Ansys Student Free Software Download

Ansys Rocky student software limitations
Windows Only
Particle Count limits DEM: maximum of 32.000 regardless of shape
SPH Elements limits: maximum of 128.000
Maximum 4 CPU Cores
No GPU availability
See the Rocky Modules available for Rocky student version at Ready-to-use Modules
See the Rocky Scripts available for Rocky student version at Ready-to-use Scripts

What would you like to do?

- Learn more about Rocky Particle Shapes and Capabilities (p. 41)
- Learn more About This Version of Rocky (p. 50)

See Also:

- Program Limitations (p. 52)
- System Requirements (p. 67)
- Best Practices (p. 68)

1.2.1. Rocky Particle Shapes and Capabilities

In Rocky, different particle shapes have different capabilities. To help you better understand what shape type and functionality combinations are possible in this version of Rocky, refer to the image and tables below.

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

Figure 1.1: Particle Shape Representations

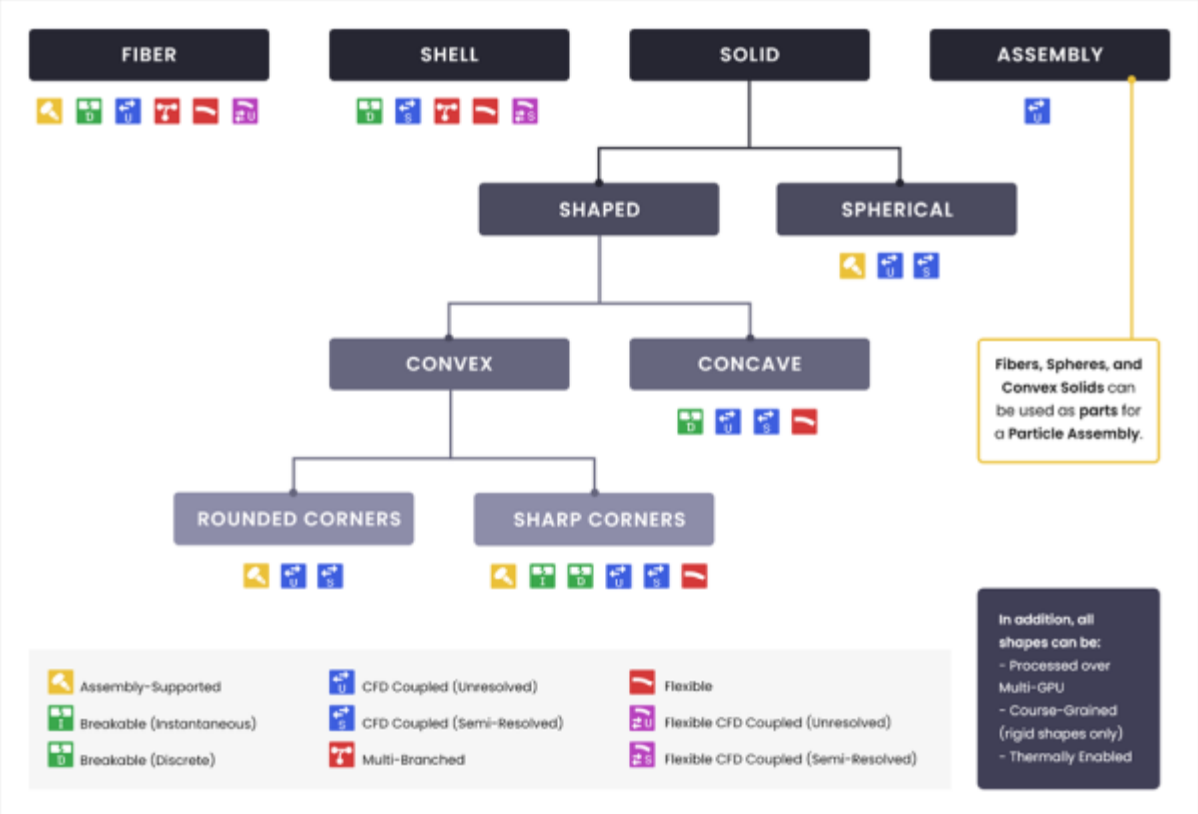


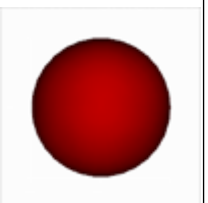
Table 1.1: Particle Shape Type and Capabilities Matrix

-	-	**Fiber***	**Shell***	**Solid Convex**	**...**	
Capability	Description	Strand-like 3D objects made by joining together linear segments of a definable thickness.	Sheet- or container-like 3D objects made by joining two-dimensional, open surfaces of a definable thickness.	Closed 3D whole shapes of full interior volume that contain no faces that form dents or hollows.	...	
-	-	-	-	Polyhedrons	Other Sharp-Cornered Shapes	Spher

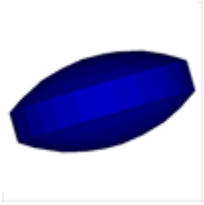
Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

-	-	Fiber*	Shell*	Solid Convex	...	
Assembly-Supported*	Shape can be used as a part (p. 1393) in an Assembly (p. 1389) shape.	✓**	✗	✓**	✓**	✓
Breakable (Instantaneous)*	Shape can be broken into fragments based upon impact and/or shear forces.	✓*	✓*	✓*	✓	
CFD Coupled (Unresolved)*	Shape is c_static/ompatible with the unresolved CFD coupling methods used to simulate the effects of fluids.	✓	✓**	✓**	✓**	✓
CFD Coupled (Semi-Resolved)	Shape is compatible with the 2-Way Fluent Semi-Resolved CFD coupling method used to simulate the effects of fluids.	✗	✓	✓**	✓**	✓
Flexible*	Shape can be composed of multiple elements that enable it to bend and flex. (See also Table 2, below.)	✓*	✓*	✓*	✗	
Multi-Branched*	Shape can include one or more bifurcating points where a segment or surface divides into a separate branch or fork.	✓	✓	n/a	n/a	n
Breakable (Discrete)	The joints between	✓	✓	✓	✓	

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

-	-	Fiber*	Shell*	Solid Convex	...	
	Multi-Element (flexible) particles can be broken into individual elements.					
Intra-particle Collision Statistics Supported	Shape supp_static/orts the gathering of collision data, which can be displayed on the surface of a representative particle.	✗**	⊗	✓	⊗	⊗
Default Shapes	Straight Fiber 	⊗	Polyhedron 	Briquette, Faceted Cylinder  	Sphere 	Spher Spher Sp Polyh   
Rocky comes with default shapes that you can use out-of-the-box.						

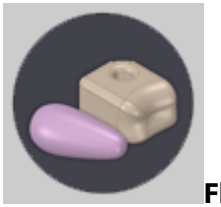
Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

-	-	**Fiber***	**Shell***	**Solid Convex**	**...**	
Custom (Imported) Shapes*	Custom Fiber 	Custom Shell 	Custom Polyhedron (Convex) 	n/a	n/a	n
Rocky enables you to define and import your own custom shapes.	Examples: Hay, hair, noodle	Examples: Snack chip, open box, sheet of paper	Examples: Tablet, cube			
Assembly Shapes	Example Fiber Assembly 	n/a	Example Custom Polyhedron (Convex) Assembly 	Example Default Shape Assembly 	n/a	n
Rocky enables you to combine several default or custom (imported) shapes together to make all new shapes.	Lollipop shape made from a Straight Fiber and a Sphere shape		Concave bolt shape made from two custom (imported) convex shapes	Snowman shape made from two Spheres, a Sphero-Polygon, and a Faceted Cylinder shape		

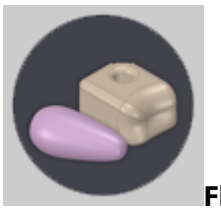
Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

** See Table 2 below for further limitations on using Flexible particles in combination with other capabilities.

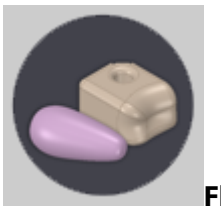
Table 1.2: Flexible Particle Capabilities (Rocky Professional and Rocky Academic Only)

Capability	Description	 **Flexible Fiber**	 **Flexible Shell**	 **Flexible Polyhedrons**	
Capability	**Description**	**Flexible, strand-like 3D objects made by joining together linear segments of a definable thickness.**	**Flexible, sheet- or container-like 3D objects made by joining two-dimensional, open surfaces of a definable thickness.**	**Solid Convex** **Flexible, closed 3D whole shapes of full interior volume that contain no faces that form dents or hollows**	**Solid Concave** **Flexible closed 3D whole or perforat shapes of full interior volume that contain at least some faces that form dents or hollows**
Multi-Branched	Shape can include one or more bifurcating points where a segment or surface divides into a separate branch or fork.	✓	✓	n/a	n/a
CFD Coupled (Unresolved)	Shape is compatible	✓	✗	✗	✗

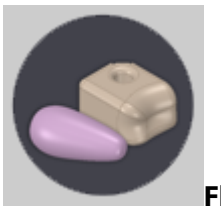
Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

Capability	Description	 **Flexible Fiber**	 **Flexible Shell**	 **Flexible Polyhedrons**	**Solid Concave**
		Flexible, strand-like 3D objects made by joining together linear segments of a definable thickness.	**Flexible, sheet- or container-like 3D objects made by joining two-dimensional, open surfaces of a definable thickness.**	**Solid Convex** **Flexible, closed 3D whole shapes of full interior volume that contain no faces that form dents or hollows**	**Flexible closed 3D whole or perforat shapes of full interior volume that contain at least some faces that form dents or hollows**
	with the unresolved CFD coupling methods used to simulate the effects of fluids.				
CFD Coupled (Semi-Resolved)	Shape is compatible with the 2-Way Fluent Semi-Resolved coupling method used to	✗	✓	✗	✗

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

Capability	Description	 **Flexible Fiber**	 **Flexible Shell**	 **Flexible Polyhedrons**	**Solid Concave**
Capability	**Description**	**Flexible, strand-like 3D objects made by joining together linear segments of a definable thickness.**	**Flexible, sheet- or container-like 3D objects made by joining two-dimensional, open surfaces of a definable thickness.**	**Solid Convex** **Flexible, closed 3D whole shapes of full interior volume that contain no faces that form dents or hollows**	**Flexible closed 3D whole or perforat shapes of full interior volume that contain at least some faces that form dents or hollows**
	simulate the effects of fluids.				
Breakable (Discrete)	The joints between Multi-Element (flexible) particles can be broken into individual elements.	✓	✓	✓	✓
Intra-particle Collision Statistics Supported	Shape supports the gathering of collision data, which can be	✓	✗	✓	✓

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

Capability	Description	 **Flexible Fiber**	 **Flexible Shell**	 **Flexible Polyhedrons**	**Solid Concave**
		Flexible, strand-like 3D objects made by joining together linear segments of a definable thickness.	**Flexible, sheet- or container-like 3D objects made by joining two-dimensional, open surfaces of a definable thickness.**	**Solid Convex** **Flexible, closed 3D whole shapes of full interior volume that contain no faces that form dents or hollows**	**Flexible closed 3D whole or perforat shapes of full interior volume that contain at least some faces that form dents or hollows**
	displayed on the surface of a representative particle.				
Coarse-Grained	Shape supports Coarse Grain Modeling.	⊗	⊗	⊗	⊗

See Also:

- About Adding and Editing Particle Sets (p. 602)
- Enable Particle Breakage Calculations (p. 397)
- Enable Thermal Modeling Calculations (p. 400)
- Set or Modify Fluid and/or Air Flow Properties (p. 757)

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

- About Intra-Particle Collision Statistics (p. 324)
- About Defining and Importing Custom Particle Shapes (p. 670)

1.2.2. PyRocky

PyRocky is part of the **PyAnsys** project, which is an Ansys initiative that aims to make available their technologies via libraries in the Python ecosystem. Each library provides clear, concise, and maintainable APIs. Useful Pythonic functions, classes, and plugins allow users to interact with targeted products and services in a high-level, object-orientated approach. The PyAnsys ecosystem refines the component-level interaction with Ansys solvers and tools, and eliminates the inconsistent and restrictive scripting environments within product installations.

Similar to PyMAPDL and PyFluent, **PyRocky** is a Python library that allows users to integrate **Rocky** in a Python environment. It is a tool for Ansys users to create, customize, and run Rocky in Python. PyRocky is in the beginner stage of development, and for now, it has the same workflow as **Rocky PrePost scripting**, but it can be run outside of Rocky, in a Python environment that best suits users.

Click here to learn more about PyRocky

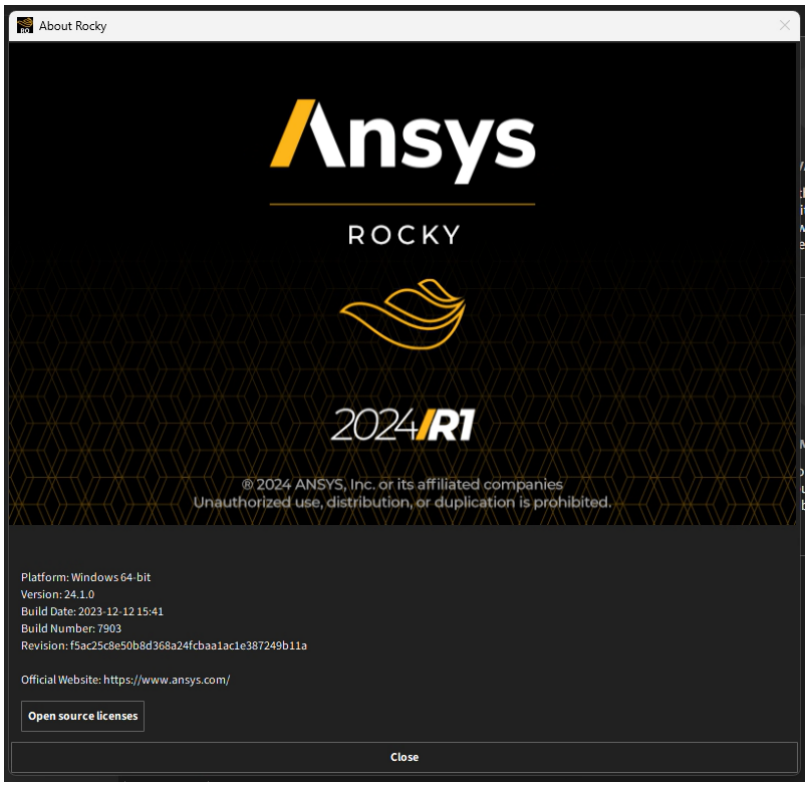
1.2.3. About This Version of Rocky

The information listed on the Rocky About screen (Figure below) includes all of the following:

- Platform
- Version
- Revision
- Release Date
- Official Website (URL)

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

Figure 1.2: About Rocky screen



In addition, the following information is provided via separate hyperlink: **Open source licenses**, which lists the open-source libraries used by Rocky, including their versions and full license text, in a separate dialog.

Tip: Both this document (Open_Source_Licenses.html) and the full contents of the application's End User License Agreement (EULA) (LICENSE.txt) are also available inside Rocky's installation folder.

To open the Rocky About screen:

1. From the Rocky program's **Help** menu, click **About**. The **About Rocky** screen opens. (See Figure above.)
2. Do any or all of the following:
   - Review the information listed on the screen.
   - Click the **Open source licenses** link.
3. When you are finished reviewing the content, click the **Close** button.

See Also:

- Package and Version Capabilities (p. 35)
- System Requirements (p. 67)

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

1.3. Program Limitations

Rocky is a dynamic program with a code base that is continually being improved upon to better serve the needs of our customers. As such, there may be some features included in this version that are not fully developed or verified. These features are provided as a working example of future capabilities but there are limits to their practical usage as described in this section.

The below matrices communicate the program limitations of major features including breakage and GPU processing (Table 1), as well as feature compatibility with the various particle shape types and models (Tables 2 and 3).

To learn more about what feature limitations might be due to your license type, or for further information regarding the known limitations of other specific Rocky features and capabilities, see the additional topics listed below.

Table 1: Compatibility Matrix for Major Rocky Features

FEATURE	Breakage	GPU and Multi GPU Processing	1-Way LBM	Thermal Modeling	1-Way Fluent	2-Way Fluent	Coarse Grain
Breakage	—	✓	✓	✓	✓	✓	✗
Energy Spectra Collection	✗	✓	✓	✓	✓	✓	✓
About Intra-Particle Collision Statistics	✗	✓	✓	✓†	✓	✓	✓
Flip Count Property	✗	✓	✓	✓	✓	✓	✓
GPU and Multi GPU Processing	✓	—	✓	✓	✓	✓	✓**
Particle Tagging, including Divisions Tagging	✓*	✓	✓	✓	✓	✓	✓
Particles Time Selection	✓*	✓	✓	✓	✓	✓	✓

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

FEATURE	Breakage	GPU and Multi GPU Processing	1-Way LBM	Thermal Modeling	1-Way Fluent	2-Way Fluent	Coarse Grain
Residence Time Property	✓*	✓	✓	✓	✓	✓	✓
1-Way LBM	✓	✓	—	✓	✗	✗	✗
Saving an In-Process Simulation for Restart	✓†	✓	✓	✓	✓	✓	✓
Extending the Duration of an In-Process or Completed Simulation	✓	✓	✓	✓	✓	✓	✓
Thermal Modeling	✓	✓	✓	—	✓	✓	✓
1-Way Fluent	✓	✓	✗	✓	—	✗	✓
2-Way Fluent	✓	✓	✗	✓	✗	—	✓
Coarse Grain Modeling	✗	✓**	✗	✓	✓	✓	—

Table 2: Compatibility Matrix for Rigid Rocky Particle Shape Types Composed of Single Elements

RIGID PARTICLE SHAPES COMPOSED OF A SINGLE ELEMENT	***Instantaneous Breakage***	**GPU and Multi GPU Processing**	**1-Way LBM**	**Thermal Modeling**	**1-Way Fluent**	**2-Way Fluent**	**Coarse Grain Modeling**
Spherical Solid Sphere	✗	✓	✓	✓	✓	✓	✓
Rounded Corner Solid *Sphero-Cylinder, Sphero-Polygon, Sphero-Polyhedron*	✗	✓	✓	✓	✓	✓	✓

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

RIGID PARTICLE SHAPES COMPOSED OF A SINGLE ELEMENT	***Instantaneous Breakage***	**GPU and Multi GPU Processing**	**1-Way LBM**	**Thermal Modeling**	**1-Way Fluent**	**2-Way Fluent**	**Coarse Grain Modeling**
Sharp Cornered Solid *Polyhedrons, Briquettes, Faceted Cylinders, and Custom Non-Concave-Shaped Polyhedron Particles*	✓	✓	✓	✓	✓	✓	✓
Concave Solid *Custom Concave-Shaped Polyhedrons*	✗	✓	✓	✓	✓	✓	✓
Fiber *Straight Fiber and Custom Fiber Particles*	✗	✓	✓	✓	✓	✓	✓
Shell *Custom Shell Particles*	✗	✓	✓	✓	✗	✗	✓

Table 3: Compatibility Matrix for Flexible Particle Shape Types Composed of Multiple Elements (Meshed)

FLEXIBLE PARTICLE SHAPES COMPOSED OF MULTIPLE ELEMENTS	***Discrete Breakage***	**GPU and Multi GPU Processing**	**1-Way LBM**	**Thermal Modeling**	**1-Way Fluent**	**2-Way Fluent**	**Coarse Grain Modeling**
Convex Polyhedron Solid Polyhedrons, and Custom Non-Concave-Shaped Polyhedron Particles	✓	✓	✓	✓	✗	✗	✗
Concave Polyhedron Solid *Custom Concave-Shaped Polyhedrons*	✓	✓	✓	✓	✗	✗	✗
Fiber *Straight Fiber and Custom Fiber Particles*	✓	✓	✓	✓	✓	✓	✗
Shell *Custom Shell Particles*	✓	✓	✓	✓	✗	✗	✗

*** Limitation still applies to projects processed in Rocky versions prior to v4. To remove the limitation, re-process the project in Rocky v4 or later.**

[cross symbol] Features are compatible but do not interact.

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

**** Note that the Radl et al. Coarse Grain Model is not currently compatible with Multi GPU processing.**

See Also:

- Package and Version Capabilities (p. 35)
- Physics and Force Limitations (p. 55)
- Geometry Limitations (p. 58)
- Particle and Input Limitations (p. 59)
- Materials and Interactions Limitations (p. 62)
- CFD Coupling Limitations (p. 63)
- Display and Processing Limitations (p. 66)
- System Requirements (p. 67)
- Best Practices (p. 68)

1.3.1. Physics and Force Limitations

The below list defines the various limitations and restrictions for setting Physics components in Rocky.

MOMENTUM SETTINGS LIMITATIONS

- Each simulation must have a Normal Force and Tangential Force defined; an Adhesive Force is not required.

Also, as specified in Table 1 below:

- The Rolling Resistance Model of Type C: Linear Spring Rolling Limit does not work with the Tangential Force of Coulomb Limit.
- The Tangential Force of Mindlin-Deresiewicz works only with the Normal Force of Hertzian Spring Dashpot.
- When the Tangential Force of Mindlin-Deresiewicz is being used in a project with shaped particles (p. 1395), it is recommended that **Rolling Resistance** be set to zero (0) for any shaped Particle sets (p. 1393) due to the fact that doing so could lead to an unstable Angle of Repose. (See also About Adding and Editing Particle Sets (p. 602).)
- The Adhesive Force of JKR works only with the Normal Force of Hertzian Spring Dashpot.
- Both types of Rolling Resistance Models are incompatible with Concave (p. 1390) Custom Polyhedron (Solid (p. 1395)) particle shapes.

Table 1: Compatibility Matrix for Rocky Momentum Models

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

MODEL	*Rolling Resistance Model: Type A: Constant Moment*	*Rolling Resistance Model: Type C: Linear Spring Rolling Limit*	*Normal Force: Hysteretic Linear Spring*	*Normal Force: Linear Spring Dashpot*	*Normal Force: Hertzian Spring Dashpot*	*Adhesive Force: Constant*	*Adhesive Force: Linear*	*Adhesive Force: JKR*	*Tangential Force: Linear Spring Coulomb Limit*	*Ta Fo Co Li*
Rolling Resistance Model: Type A: Constant Moment	—	—	✓	✓	✓	✓	✓	✓	✓	✓
Rolling Resistance Model: Type C: Linear Spring Rolling Limit	—	—	✓	✓	✓	✓	✓	✓	✓	✗
Normal Force: Hysteretic Linear Spring	✓	✓	—	—	—	✓	✓	✗	✓	✓
Normal Force: Linear Spring Dashpot	✓	✓	—	—	—	✓	✓	✗	✓	✓
Normal Force: Hertzian Spring Dashpot	✓	✓	—	—	—	✓	✓	✓	✓	✓
Adhesive Force: Constant	✓	✓	✓	✓	✓	—	—	—	✓	✓
Adhesive Force: Linear	✓	✓	✓	✓	✓	—	—	—	✓	✓
Adhesive Force: JKR	✓	✓	✗	✗	✓	—	—	—	✓	✓
Tangential Force: Linear Spring Coulomb Limit	✓	✓	✓	✓	✓	✓	✓	✓	—	—
Tangential Force:	✓	✗	✓	✓	✓	✓	✓	✓	—	—

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

MODEL	*Rolling Resistance Model: Type A: Constant Moment*	*Rolling Resistance Model: Type C: Linear Spring Rolling Limit*	*Normal Force: Hysteretic Linear Spring*	*Normal Force: Linear Spring Dashpot*	*Normal Force: Hertzian Spring Dashpot*	*Adhesive Force: Constant*	*Adhesive Force: Linear*	*Adhesive Force: JKR*	*Tangential Force: Linear Spring Coulomb Limit*	*T F C Li*
Coulomb Limit										
Tangential Force: Mindlin-Deresiewicz	✓*	✓*	✗	✗	✓	✓	✓	✓	—	—

* For this combination of models, setting Rolling Resistance values are recommended only for spherical Particle sets. For shaped Particle sets, it is recommended that Rolling Resistance be set to zero (0).

THERMAL MODELING LIMITATIONS

The 2-Way Fluent Semi-Resolved method does not currently support Thermal Modeling. (See also About Using the 2-Way Fluent Semi-Resolved Method (p. 831).)

Coarse-Graining Settings Limitations

Coarse Grain Modeling (CGM) is incompatible with all of the following features and functionality:

- Particle breakage.
- Particles composed of multiple elements (also known as Meshed (p. 1392)). This includes flexible Fibers (p. 1391), Shells (p. 1395), and Solids (p. 1395).
- The Velocity Dependent Restitution Model.
- Non-drag fluid forces. This includes torque, virtual mass, and lift forces.
- 1-Way LBM CFD Coupling.

In addition, the Radl et al. Coarse Grain Model is incompatible with Multi GPU processing.

Important:

When CGM is enabled, the results might differ significantly from those without CGM. This is due to an uncertainty that is incorporated within the numerical results, for the CGM is a mathematical manipulation to artificially increase the size of the particles. This kind of mathematical modeling needs to be used with care and evaluating if the results make physical sense.

See Also:

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

- About Physics Parameters (p. 417)
- Program Limitations (p. 52)

1.3.2. Geometry Limitations

The below list defines the various limitations and restrictions for setting Geometry components in Rocky.

ALL GEOMETRIES

- Geometries cannot have duplicate names. (I.e., two different geometry components cannot have the same name.)

IMPORTED GEOMETRIES ONLY

- Imported geometries with surface wear enabled have all of the following limitations:
  - They cannot have a **Triangle Size** that is too large relative to the geometry dimensions.
  - They cannot have a Motion Frame assigned with the **Keep in Place** option enabled (set as **Global** or **Local**). (Surface wear is incompatible with any motions without displacement.)
- Imported geometries with Motion Frames assigned have all of the following limitations:
  - A Motion Frame with a Free Body Translation or Free Body Rotation motion(s) must be assigned to an imported geometry; having a Motion Frame with a Free Body motion but no geometry assigned to it is not supported.
  - For any given Motion Frame, the time intervals for two or more of the same kind of Free Body motions (Translation or Rotation) that share common directions (X and XY, for example) cannot overlap. This means that a Free Body Translation motion in the X direction cannot occur at the same time as a second Free Body Translation motion in the XY direction, but can occur at the same time as Free Body Rotation motion, or at the same time as a Free Body Translation motion on a different axis (Y, for example).

    Note:

    Note: The time intervals are open at the end. For example, the intervals (0s, 10s) and (9s, 20s) overlap but the intervals (0s, 10s) and (10s, 20s) do *not* overlap because the second motion starts at exactly the stop time of the first.

  - Imported geometries with Geometry Replication enabled must have a Motion Frame assigned with at least one of the following motions: Translation, Rotation, Periodic Translation (Vibration), or Periodic Rotation (Pendulum). (This means that geometry replication features are not compatible with free body motions.)

See Also:

- About Adding Default Geometry Components (p. 483)
- About Creating and Applying Motion Frames (p. 533)

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

• Program Limitations (p. 52)

1.3.3. Particle and Inlet Limitations

The below list defines the various limitations and restrictions for setting Particles and Inlets in Rocky.

GENERAL LIMITATIONS FOR PARTICLES AND PARTICLE INLETS

- There must be at least one particle set (p. 1393) and Inlet defined in the simulation.
- A simulation may have both the Thermal Model enabled and have collections for Intra-particle Collision Statistics enabled, but these two features do not interact with each other. In these cases, Temperature properties can be analyzed for the Particle set as a whole but cannot be viewed in the Particles Details window on a single representative particle. (See also Enable Thermal Modeling Calculations (p. 400) and About Intra-Particle Collision Statistics (p. 324).)
- Particles with breakage enabled are incompatible with Coarse Grain Modeling. (See also About Physics Parameters (p. 417).)

GENERAL PARTICLE SHAPE LIMITATIONS

- A particle's horizontal and vertical aspect ratios cannot both be simultaneously bigger than 1.0.
- Briquette particles must have a number of corners that is a multiple of 4.
- Faceted Cylinder particles must have an even number of corners.
- The cumulative size distribution list for Particles must be in descending order for both the sizes (represented by either the **Size**, **Diameter**, or **Scale Factor** column) and the percentages, and the percentages must add up to 100%.
- Rigid concave particles do not support editing Rolling Resistance values. As a result, Rolling Resistance will always be set to zero (0) for these shapes.
- Rigid Shell (p. 1395) and Fiber (p. 1391) particle shapes composed of single elements do not support Intra-particle Collision Statistics.
- Frozen Fibers (p. 1391) work only when defined with multiple elements. (The Particle set's (p. 1393) **Composition** parameter must be set to **Multiple Elements**.)
- Concave (p. 1390) Custom Polyhedron (Solid (p. 1395)) particle shapes are incompatible with both types of Rolling Resistance Models.

ASSEMBLY PARTICLE SHAPE LIMITATIONS

- There can be only 64 individual parts (p. 1393) per Assembly (p. 1389) shape.
- The individual parts (p. 1393) making up an Assembly particle shape can come only from Particle sets (p. 1393) defined as single element (rigid), non-breakable Solid (p. 1395) or Straight Fiber (p. 1391) shapes. The following types of Particle sets cannot be used as parts of an Assembly shape:
  - Particle shapes that have Multiple Elements defined (flexible particles).
  - Concave (p. 1390) shapes that were imported as a Custom Polyhedron.

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

  - Shell (p. 1395) particle shapes.
  - Custom Fiber (p. 1391) shapes that you have imported. (Default Straight Fiber shapes are supported, however.)

- When used in CFD Coupling cases, Assembly particle shapes are not compatible with the Ganser Drag Law, and can also not be used in 2-Way Fluent Semi-Resolved cases.
- As Assembly particle shape does not support breakage models nor can be made of multiple elements (flexibility).
- The parts making up an Assembly particle will only be shown in one color for the whole Assembly.

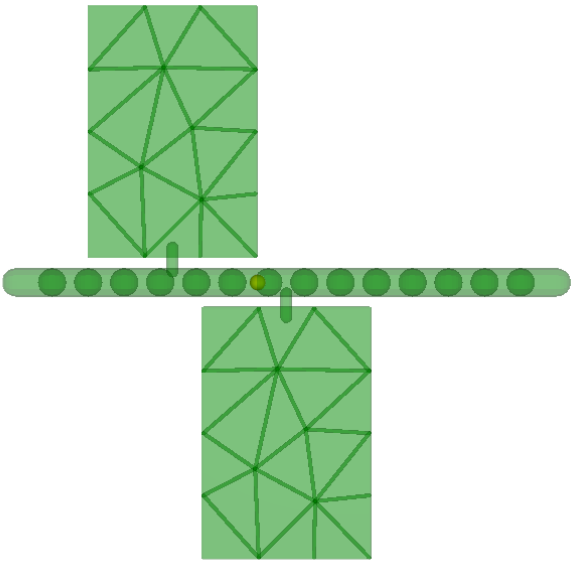
Particle Assembly Beta Feature: Flexible Fibers and Shells and Discrete Breakage

When the **Experimental (Beta) Features** checkbox is enabled on the **Options | Preferences** dialog (see also About Setting Global Preferences (p. 227)), this allows you to use flexible fibers and shells, as well as Discrete Breakage with Particle Assembly.

As it is a beta, this functionality has some limitations. For instance, it is not possible to create a joint between two rigid compound parts, and you have to make sure that there is at least one flexible element between each pair of flexible elements attached to a rigid part, in all directions. The figure below illustrates two different scenarios. In Figure 1.3: Example of correct condition for Flexible Particle Assembly (p. 60) you can see a scenario where we have at least one flexible element between two rigid parts, which is correct for the simulation. In the other hand, in Figure 1.4: Example of Inappropriate condition for Flexible Particle Assembly (p. 61) we have an example of an inappropriate condition. You can see that even though there is at least one flexible element between the rigid parts, Rocky considers a 10% upscaling tolerance that you have to take into account.

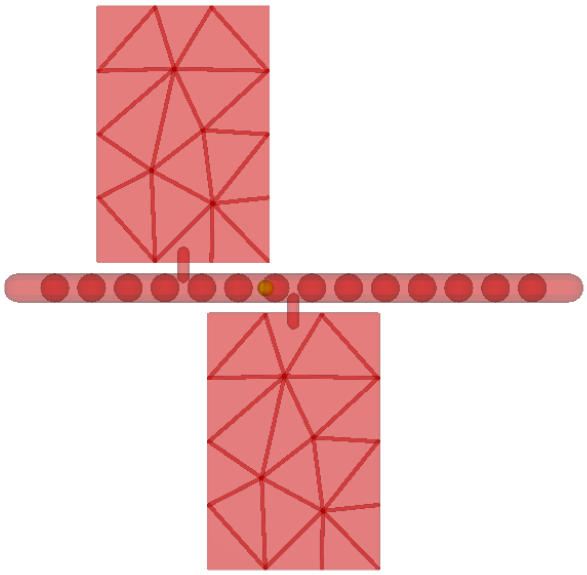
Note that if you have larger elements when compared to the rigid parts, the rigid parts upscaling will be less significative.

Figure 1.3: Example of correct condition for Flexible Particle Assembly



Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

Figure 1.4: Example of Inappropriate condition for Flexible Particle Assembly



MULTI-ELEMENT (FLEXIBLE AND/OR DISCRETE-BREAKABLE) PARTICLE SHAPE LIMITATIONS

- All Shell (p. 1395) and Fiber (p. 1391) particle shapes can be composed of Multiple Elements, which is also known as meshed (p. 1392). In the Solid (p. 1395) particle shape category, however, only Polyhedron and Custom Polyhedron shapes support meshing. All other kinds of Solid shapes must be composed of only a single element (non-meshed).

- Flexible particles-which are composed of Multiple Elements and are also known as meshed (p. 1392) particles-can have only a single **Size** (**Diameter** or **Scale Factor**) defined per particle set (p. 1393). (This can also be known as a "monosized" particle.)

- Shell and Solid particles composed of Multiple Elements (also known as meshed (p. 1392) particles) are incompatible with Fluent CFD coupling, including the 1- and 2-Way Fluent options. The only meshed particle shapes compatible with these features are Fibers (p. 1391). (See also About Adding and Editing Particle Sets (p. 602).)

- Flexible particles-which are composed of Multiple Elements and are also known as meshed (p. 1392) particles- do not support editing Rolling Resistance values. As a result, Rolling Resistance will always be set to zero (0) for these shapes.

- Flexible particles-which are composed of Multiple Elements and are also known as meshed (p. 1392) particles- also do not support editing Enable Rotations settings. As a result, Enable Rotations will always be turned on (enabled) for these shapes.

- Flexible particles-which are composed of Multiple Elements and are also known as meshed (p. 1392) particles- that are used for the analysis of any particle-related collisions statistics (see also About Inter-group Collision Statistics (p. 319), About Intra-Particle Collision Statistics (p. 324), or About Inter-particle Collision Statistics (p. 331)), will either ignore the effects of upscaling or will not include some properties and curves even when the **Meshed Particles Upscaling** checkbox is enabled. (See also About Meshed Particles Upscaling (p. 1033).)

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

- Plasticity models are available only for Shell (p. 1395) and Fiber (p. 1391) particle shapes that are composed of Multiple Elements, which is also known as meshed (p. 1392). In this version of Rocky, plasticity models are not available for flexible Solid (p. 1395) particles.
- Particles composed of multiple elements (also known as meshed (p. 1392)) are incompatible with Coarse Grain Modeling. This includes flexible Fibers (p. 1391), Shells (p. 1395), and Solids (p. 1395).
- Particles composed of multiple elements (also known as meshed (p. 1392)) are incompatible with the Flip Count property for Particles.

INLET LIMITATIONS

- Every continuous injection Inlet must have its entry point set.
- Every entry for a Volumetric Inlet must have a positive Mass value.
- At least one Particle set or fluid within the continuous injection Inlet must have a mass flow rate value greater than 0 t/h.
- Periodic continuous injections are incompatible with disabling the **Release Particles Without Overlap Checks** setting.
- Volumetric Inlets require a minimum of 3 particles in order to create the "Seed". (You are not able to create a Volumetric Inlet with less than 3 particles.)
- The three-way combination of a (1) Particle Custom Inlet that uses (2) frozen Fibers (p. 1391) and also has a (3) Motion Frame assigned to it is, as a specific combination, incompatible with both Cartesian and Cylindrical Periodic Domains. (See also About Domain Settings Parameters (p. 452).)

See Also:

- About Adding and Editing Particle Sets (p. 602)
- Rocky Particle Shapes and Capabilities (p. 41)
- Program Limitations (p. 52)

1.3.4. Materials and Interactions Limitations

The below list defines the various limitations and restrictions for setting Materials and its Interactions, in Rocky.

MATERIAL LIMITATIONS

- (Recommendation) Solid Materials should have a **Poisson ratio** in the recommended range (0.0 to 0.5).
- Every entity that has solid materials (conveyors, geometries, and particles) must have a Solid Material set.

MATERIALS INTERACTIONS LIMITATIONS

- A Materials Interactions' coefficient of restitution must be in the predefined range (0.1 to 1.0).

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

- When using the Linear **Adhesive Force** (see also About Physics Parameters (p. 417)) for Materials Interactions, the **Stiffness fraction** between two materials must be in the range of [0.0 to 1.0).
- When using the Constant **Adhesive Force** (see also About Physics Parameters (p. 417)) for Materials Interactions, the **Force fraction** between two materials must not be negative.

See Also:

- About Modifying Material Compositions (p. 591)
- About Modifying Materials Interactions and Adhesion Values (p. 597)
- Program Limitations (p. 52)

1.3.5. SPH Limitations

The below list defines the limitations and restrictions for setting a fluid flow simulation using the SPH functionality in Rocky.

GENERAL LIMITATIONS FOR SPH SIMULATIONS IN ROCKY

- Only one phase of SPH elements is supported per simulation.
- Shell particles will only interact with SPH elements if the SPH element size is at least 4 times greater than the Shell particle thickness.

See Also:

- About Adding and Editing Particle Sets (p. 602)
- About Adding and Editing Inlets and Outlets (p. 708)
- About SPH Parameters (p. 436)
- Program Limitations (p. 52)

1.3.6. Ansys and CFD Coupling Limitations

The below lists define the various limitations and restrictions for using the 1-Way LBM method, the 1-Way Constant Coupling Method, and any Ansys Fluent simulations that are coupled 1-Way or 2-Way with Rocky.

1-WAY LBM COUPLING LIMITATIONS

- Shell (p. 1395) and Solid (p. 1395) particles composed of Multiple Elements (p. 1390) (also known as meshed (p. 1392), a model which can enable flexible and/or discrete-breakable particles) are incompatible with 1-Way LBM. The only flexible particle shapes compatible with this feature are Fibers (p. 1391). (See also About Adding and Editing Particle Sets (p. 602).)
- The Air Flow boundary box that you define through the three **Coordinate Limits** fields will interact with the flow of air just as would a solid wall geometry. For this reason, it is recommended that you increase the size of the air flow boundary box well beyond the area where you expect to see

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

air flow interactions. (Note that increasing your Air Flow boundary size will also increase the processing time due to the additional amount of air flow cells that will need to be calculated.)

- Setting a smaller Air Flow **Cell Size** than your largest particle can result in air flow vectors incorrectly appearing outside of the geometries that are supposed to contain them. For this reason, it is recommended that you select an Air Flow **Cell Size** at least as large as your largest particle size.

- The 1-Way LBM coupling method is incompatible with Coarse Grain Modeling. (See also About Physics Parameters (p. 417).)

1-WAY CONSTANT COUPLING LIMITATIONS

- When combined with Coarse Grain models, only drag forces are considered. This means that other non-drag fluid force models, such as virtual mass and lift, are incompatible with Coarse Grain modeling.

- Shell (p. 1395) and Solid (p. 1395) particles composed of Multiple Elements (p. 1390) (also known as meshed (p. 1392), a model which can enable flexible and/or discrete-breakable particles) are incompatible with the 1-Way Constant method. The only flexible particle shapes compatible with this method are Fibers (p. 1391). (See also About Adding and Editing Particle Sets (p. 602).)

1-WAY AND 2-WAY FLUENT COUPLING LIMITATIONS

- CFD coupling simulations with Fluent (for both 1-Way and 2-Way) must have the same Thermal Model setting in both Rocky and Fluent. That is, both Rocky and Fluent must have thermal modeling either enabled or disabled. (See also Enable Thermal Modeling Calculations (p. 400).)

- When combined with Coarse Grain models, only drag forces are considered. This means that other non-drag fluid force models, such as virtual mass and lift, are incompatible with Coarse Grain modeling.

- Shell (p. 1395) and Solid (p. 1395) particles composed of Multiple Elements (p. 1390) (also known as meshed (p. 1392), a model which can enable flexible and/or discrete-breakable particles) are incompatible with the 1-Way Fluent and 2-Way Fluent methods. The only flexible particle shapes compatible with these methods are Fibers (p. 1391). (See also About Adding and Editing Particle Sets (p. 602).)

- In addition, Shell (p. 1395) particles composed of Single Elements (also known as a "rigid" particle shape) are incompatible with both the 1-Way Fluent and 2-Way Fluent coupling methods.

- The Ganser Drag Law is not supported for Assembly (p. 1389) particle shapes.

See also the **Coupling limitations** section within the **CFD Coupling Technical Manual**. (From the Rocky **Help** menu, point to **Manuals** and then click **CFD Coupling Technical Manual**.)

2-WAY FLUENT COUPLING LIMITATIONS

- 2-Way Fluent coupled simulations with Remesh are not compatible with Sub-Stepping method.

See also the **Coupling limitations** section within the **CFD Coupling Technical Manual**. (From the Rocky **Help** menu, point to **Manuals** and then click **CFD Coupling Technical Manual**.)

2-WAY FLUENT AND 2-WAY FLUENT SEMI-RESOLVED COUPLING LIMITATIONS

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

- Multiphase (p. 1392) 2-Way Fluent simulations are not compatible with the **Multi-Fluid VOF Model** in Fluent. When setting up your Fluent case, ensure that all **Eulerian Parameters** are cleared on the **Mutiphase Model** dialog.

2-WAY FLUENT SEMI-RESOLVED COUPLING LIMITATIONS

- 2-Way Fluent Semi-Resolved simulations are not compatible with thermal modeling. (See also Enable Thermal Modeling Calculations (p. 400).)
- 2-Way Fluent Semi-Resolved simulations work only with Spheres, Shells (either rigid or flexible), or Custom Imported Polyhedrons (rigid only). This method is not compatible with Fibers (neither rigid nor flexible), default Solids, nor flexible Custom Imported Polyhedron particle shapes. (See also Rocky Particle Shapes and Capabilities (p. 41).)
- The 2-Way Fluent Semi-Resolved method will not provide fluid-related properties for post-processing in Rocky. (See also About Properties (p. 933).)
- 2-Way Fluent Semi-Resolved simulations do not work with the CFD Coupling Particle Statistics Module. (See also About the CFD Coupling Particle Statistics Module (p. 873).)
- 2-Way Fluent Semi-Resolved simulations are not compatible with turbulent fluid flows.
- 2-Way Fluent Semi-Resolved simulations are not compatible with Assembly (p. 1389) particle shapes.

ANSYS WORKBENCH LIMITATIONS

- Ansys Workbench integration with Rocky works only when linked with a geometry file from Ansys SpaceClaim. This means that using Rocky through Workbench will not work with the following scenarios:
  - No imported geometries. (I.e., Default Rocky conveyors only.)
  - Geometries imported into Rocky that are not linked through SpaceClaim.

Note:

The Rocky-Workbench coupling is not available on Linux OS because of dependencies on the CAD reading file process by SpaceClaim.

(See also About Rocky and Ansys Workbench Integration (p. 144).)

See Also:

- The **CFD Coupling Technical Manual** document. (From the Rocky **Help** menu, point to **Manuals** and then click **CFD Coupling Technical Manual**.)
- About Using the 1-Way LBM Method (p. 771)
- About Using the 1-Way Constant Method (p. 763)
- About Using the 1-Way Fluent Method (p. 775)
- About Using the 2-Way Fluent Method (p. 795)

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

- About Rocky and Ansys Workbench Integration (p. 144)

1.3.7. Display and Processing Limitations

The below list defines the various limitations and restrictions for displaying Rocky and processing your simulations.

- If the detected OpenGL version is older than 2.1, 3D rendering will be slow. This can happen, for instance, when accessing Rocky via Remote Desktop connections.
- GPU simulations require a CUDA-enabled GPU.
- If want to save for restart purposes a partially processed simulation with breakage enabled, be aware that any fragments (p. 1393) that were in the simulation at the time you save the copy will not be present in the copied version due to incompatibilities with the features.
- If you stopped a simulation that was in the middle of processing and later want to resume processing it using a newer version of Rocky, you may only resume if the simulation was last processed in the same major version of the product as the version you are using now. For example, if you started processing your simulation in version 4.1.0, you may resume processing it in any 4.1.x version released thereafter, but may not resume it in any version of the next major release of 4.2.x, and so on.
- Multi GPU processing is incompatible with the Radl et al. Coarse Grain Model.

See Also:

- System Requirements (p. 67)
- Program Limitations (p. 52)
- About Physics Parameters (p. 417)
- About Starting a Simulation (p. 908)

1.3.8. Module Limitations

The below list defines the various limitations and restrictions for the installed-by-default modules that come with Rocky.

COLLISION STATISTICS LIMITATIONS

- With the exception of some Inter-particle Collision Statistics properties (see also About Inter-particle Collision Statistics (p. 331)), when analyzing the resulting properties or curves of most particle-related collisions statistics (see also About Inter-group Collision Statistics (p. 319) or About Intra-Particle Collision Statistics (p. 324)), the particle upscaling effects for Flexible particles-which are composed of Multiple Elements and are also known as meshed (p. 1392) particles- will be ignored even when the **Meshed Particles Upscaling** checkbox is enabled. (See also About Meshed Particles Upscaling (p. 1033).)

CFD COUPLING PARTICLE STATISTICS LIMITATIONS

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

The 2-Way Fluent Semi-Resolved method does not currently support the CFD Coupling Particle Statistics Module. (See also About Using the 2-Way Fluent Semi-Resolved Method (p. 831).)

See Also:

- About Physics Parameters (p. 417)
- About Modules Parameters (p. 431)
- Program Limitations (p. 52)
- About Setting Global Preferences (p. 227)

1.3.9. Known Limitations in Ansys Rocky

The following sections list the limitations known to exist in this version.

SOLVER

- In some 2-Way Fluent coupled cases, when setting a value greater than 1 for the Fluent Outputs Multiplier parameter, some instabilities can be observed for pressure and mass flow rate. (1053441)
- In some 2-Way Fluent coupled cases, when assigning a large number of CPUs for Fluent, considerable differences in the mass imbalance can be observed. (1083867)

1.4. System Requirements

Rocky has been designed to be used in conjunction with CAD (p. 1389) design tools, such as Ansys Discovery; CFD (p. 1390) tools, such as Ansys Fluent; and FEA (p. 1391) tools, such as Ansys Mechanical. As a result, Rocky will have similar system requirements. Please see Ansys's current plan for supporting different operating systems with Rocky software, visit the System Requirements page on Ansys website.

ROCKY COMPATIBILITY WITH OTHER ANSYS PRODUCTS

This version of Rocky can be coupled with several other currently verison of Ansys products including:

- Ansys Fluent®;
- Ansys Mechanical™;
- Ansys Workbench™;
- Ansys Minerva®;
- Ansys Motion™;
- Ansys optiSLang®.
- Ansys EnSight™;
- Ansys Discovery™;
- Ansys SpaceClaim®.

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

1.5. Best Practices

The engineers who developed and tested Rocky have compiled the following set of best practices to help you get the most out of using the software. These recommendations are best used in conjunction with the user assistance content in this manual.

What would you like to learn more about?

- Best Practices for Installing a New Version of Rocky (p. 68)
- Best Practices for Faster Processing (p. 68)
- Best Practices for Creating Geometries for Import (p. 70)
- Best Practices for Setting up Geometries (p. 70)
- Best Practices for Setting Up Particles (p. 72)

See Also:

- Software Overview (p. 31)

1.5.1. Best Practices for Installing a New Version of Rocky

New versions of the Rocky software are released on a regular basis, which give you access to various improvements, fixes, and additions to the software. There are some steps you can take to help ensure success with the process of installing or upgrading to a new version of Rocky, including the following:

- **Manually update the driver software for your GPU cards, even if the device manager shows that it is up-to-date.** Oftentimes, the NVIDIA automatic update system does not properly install the latest drivers, and having older drivers may prevent Rocky from working properly. Visit the NVIDIA website. and manually update your GPU driver to the latest that supports the CUDA 12.8 toolkit or higher.
- **Review the Release Notes.** This is displayed automatically whenever a new version of the Rocky software is available for download, and lists all the new features and functionality that have changed since the last version. Knowing what has changed can help you locate the feature in the product and learn more about it in Help (this document). You can also view the Release Notes at any time from within the product. (From the Rocky **Help** menu, click **Release Notes**.)

See Also:

- Set Up a New GPU Card (p. 404)

1.5.2. Best Practices for Faster Processing

The amount of time it takes for Rocky to process a simulation depends upon many factors, including but not limited to:

- The processing speed of the computer Rocky is running upon
- Whether you are using CPU processing (p. 1390), GPU processing (p. 1392), or Multi GPU processing capabilities

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

- The number, size, and complexity of the particles (p. 1393) being simulated
- The size of the simulation time step (p. 1395), which is defined by particle size, density, Young's Modulus, and restitution coefficients
- The size of the geometries (p. 1391)
- The size of the boundary (p. 1389) triangles (p. 1395)
- The simulation time (in seconds)
- Whether the simulation files are being saved over a network

While there are no hard-and-fast rules regarding processing speed in Rocky, there are some steps you can take to speed up your processing time, including:

- Use GPU or Multi GPU processing, and/or increase your processing power.
- Reduce your particle stiffness, which is based upon the Young's Modulus and particle size. The default Young's Modulus of $1x10^8 Pa$ Pa can be safely reduced to $1x10^6 Pa$ - $1x10^7 Pa$ Pa for narrow size distributions (1:3 or less). This should result in a 3-10x increase in time step , which will speed up your simulations. (See also About Modifying Material Compositions (p. 591).)
- Use the Numerical Softening Factor to increase the size of your time steps. (See also About Physics Parameters (p. 417).)
- Select a simpler, easier-to-calculate model for the Normal Force and/or Tangential Force option. (See also About Physics Parameters (p. 417).)
- Instead of composing your particle shapes from Multiple Elements (p. 1390) (also known as meshing (p. 1392)), choose a Single Element **Composition**. (See also About Adding and Editing Particle Sets (p. 602).)
- Increase your minimum particle size.
- Reduce the complexity (sides, corners, size distributions) of your particles.
- Shorten the simulation length.
- Increase the size of your geometry triangles. The default value of 0.1 is recommended for all boundaries where you want to see wear data.
- Run and save Rocky files locally; avoid processing and saving Rocky simulations over a network.
- Run only one simulation at a time on any one machine.

To gain an understanding of how much faster GPU processing could be as compared with CPU processing, review the GPU Buying Guide Page: Rocky GPU Buying Guide.

See Also:

- Best Practices for Creating Geometries for Import (p. 70)
- Best Practices for Setting up Geometries (p. 70)

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

- Best Practices for Setting Up Particles (p. 72)

1.5.3. Best Practices for Creating Geometries for Import

Use the following recommendations when using a CAD program (p. 1389) to create geometries geometries (p. 1391) for importing into Rocky:

- Orient your design around the origin. For example, when designing conveyor chutes, place the center of the head pulley at (0,0,0), and build your other components around it.
- Include the conveyors in your design but do not import them into Rocky. It is easier to use the conveyors included with Rocky and only import your chutes or other geometries. Use the conveyor dimensions in your CAD design to size and position the Rocky conveyors accordingly.
- Make your design as realistic as possible. Use actual dimensions whenever possible.
- Remove any "hidden" boundaries (p. 1389) before saving the file you want to import. Some hidden geometries might be imported into Rocky.
- Color geometry components differently to make it easier to distinguish components before and after import.

See Also:

- Best Practices for Faster Processing (p. 68)
- Best Practices for Setting up Geometries (p. 70)
- Best Practices for Setting Up Particles (p. 72)

1.5.4. Best Practices for Setting up Geometries

Use the following recommendations when setting up geometries (p. 1391) in Rocky:

- Only import your non-conveyor geometries into Rocky (i.e., your chute or mill). Use the conveyor designs included within Rocky for your conveyor components.
- Make the conveyor settings as realistic as possible. Any error or shortcut taken in setting up geometries will result in false data.
- Ensure that the triangle (p. 1395) size of your feed (p. 1391) and receiving conveyors (p. 1394) are small enough to avoid deformation of the geometries. Larger triangle sizes (1m for example) might change the overall shape of the conveyor, making your simulation less realistic. It is recommended that the default value of 0.1 be used.

Note:

Triangle size will not affect the shape of imported boundaries (p. 1392); just the default conveyors included with Rocky.

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

- Ensure that your inlet is large enough to allow your largest particle to pass through, but still slightly smaller than your geometry opening. This will help prevent particles from getting "stuck".
- When setting up multiple inlets (p. 1392), ensure that your inlet locations and Start and Stop Times are configured in such a way as to avoid Particle sets from overlapping during the simulation. An inlet will delay releasing its particles until no other particles from other Particle sets are in the way.

  Note:

  This step is unnecessary if **Release Particles without Overlap Check** is selected on the **Solver | General** tab. (From the **Data** panel, click **Solver** and then from the **Data Editors** panel, select **General** on the **Solver** tab.)

- When locating an inlet or default conveyor (p. 1390), first sketch its size and location in a CAD program and then use the resulting coordinates to properly place it within Rocky.
- Ensure that particle speed equals belt speed by the time the particles reach the pulley. Adjust the loading length of the belt to achieve the desired result.
- When setting up custom feeder boxes, ensure that there is enough vertical space between the wall of the box and the entry point of the particles (i.e. top of the feeder box or inlet) for your largest particle. If a particle hits a boundary (p. 1389) or other particle before fully entering the simulation, all particles will stop flowing.

  Note:

  This step is unnecessary if **Release Particles without Overlap Check** is selected on the **Solver | General** tab. (From the **Data** panel, click **Solver** and then from the **Data Editors** panel, select **General** on the **Solver** tab.) To prevent this from happening, do one of the following (see also in ??? (p. ?))

  – Make your inlet smaller.

  – Add a vertical lip to the top of your feeder box.

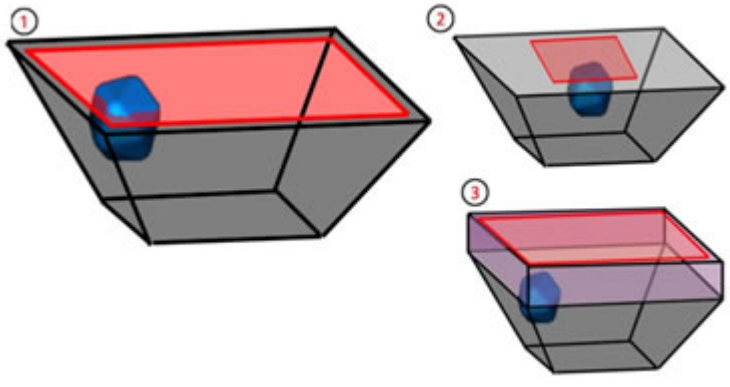


Figure 1: (1) Angled walls of feeder box prevents particle from fully entering simulation area; (2) Make the inlet smaller; (3) Add a lip to the top of the feeder box.

See Also:

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

- Best Practices for Faster Processing (p. 68)
- Best Practices for Creating Geometries for Import (p. 70)
- Best Practices for Setting Up Particles (p. 72)

1.5.5. Best Practices for Setting Up Particles

Use the following recommendations when setting up particles (p. 1393) in Rocky:

- Calibrate your particle size, shape, and interaction settings to the real-world behavior of the material you are simulating. You may wish to use the Material Wizard and/or the Calibration Suite to help you with this task.
- When setting particle friction values:
  - For rock or granular particles with rougher surfaces, set friction values to 0.6 - 0.7
  - For glass-like particles with smoother surfaces, set friction values no lower than 0.3
- When testing out a new chute design, save time by running a quick simulation with Sphere particles to ensure your boundary (p. 1389) settings are as you want them. This will take less time and will enable you to iterate on your design faster. Once your boundaries are set and verified, set up your shaped particle (p. 1395) combinations and distributions and run that as a final step.
- When using periodic Cartesian domains (previously known as "periodic boundaries"; see also About Domain Settings Parameters (p. 452)), ensure that your largest particle size is less than 0.4 times the distance between the **Min Coordinate** and **Max Coordinate** values for the periodic domain.

See Also:

- Best Practices for Faster Processing (p. 68)
- Best Practices for Creating Geometries for Import (p. 70)
- Best Practices for Setting up Geometries (p. 70)

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

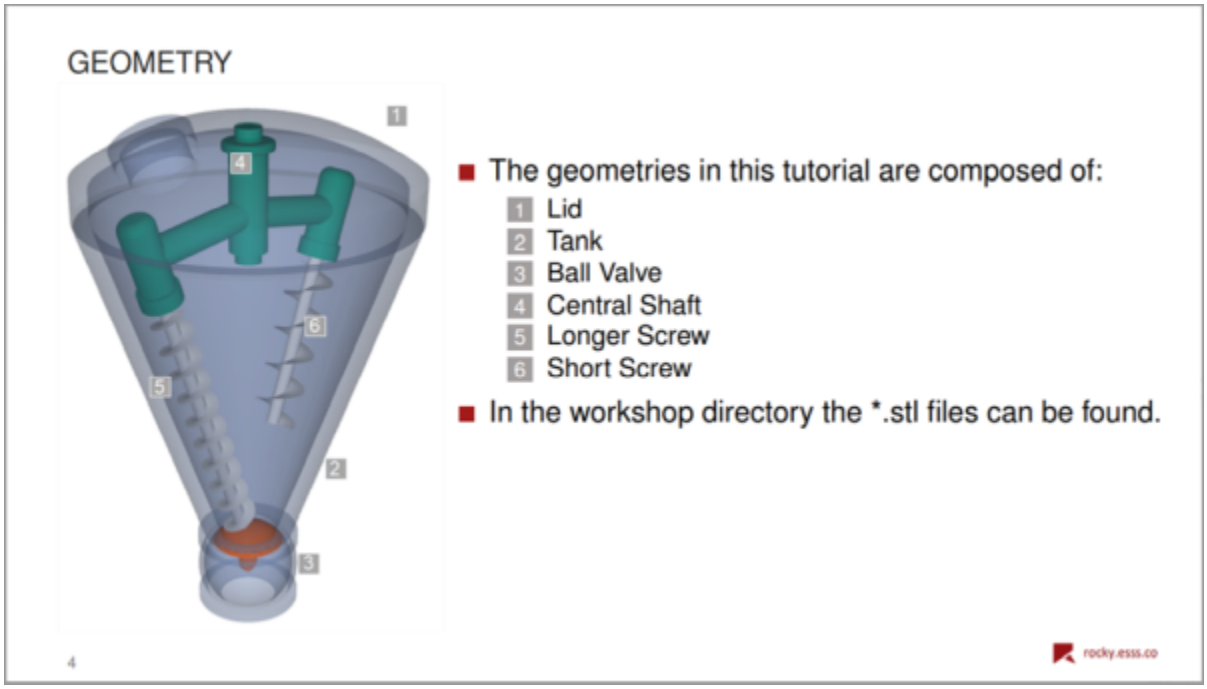
Chapter 2: Getting Started

Use the following information to get started with Rocky and gain an understanding of its various features and components.

2.1. Self-Guided Tutorials

The Rocky Tutorials provide everything you need to walk through setting up, running, and post-processing real Rocky cases at your own pace. Each Tutorial includes sample geometries and PDFs of step-by-step instructions.

Figure 2.1: Example screenshot from Rocky Tutorial 07 - Conical Dryer (Thermal Modeling) Part A: Project Setup and Processing



To gain the most benefit, complete the numbered Tutorials in the order provided, or use the Index to locate Tutorials containing specific features or tasks.

To access the Tutorials:

1. Inside the Ansys Rocky software page in Ansys Help access the Rocky Tutorial Guide.

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

2. Each chapter of the guide is a specific Tutorial, choose the one you want to learn and in its contents you will find the instructions to download the input files for that tutorial.

Note:

You can also download the Rocky Tutorial Guide in PDF along with the input files for all tutorials by clicking **Rocky Tutorial Files & PDF.**

2.2. User Interface

Ansys Rocky software is a powerful program with a flexible, hierarchical interface. One main screen gives you access to the most frequently used functions and features, and provides you with various views of the simulation.

Additional panels (p. 1393) allow you to access properties and settings to fine-tune your simulation and view statistics based upon the results.

What would you like to learn about?

- Main UI Components (p. 74)
- Hidden Features (p. 112)
- Keyboard Accessibility and Shortcuts (p. 128)
- File Types and Folders in Rocky (p. 129)

See Also:

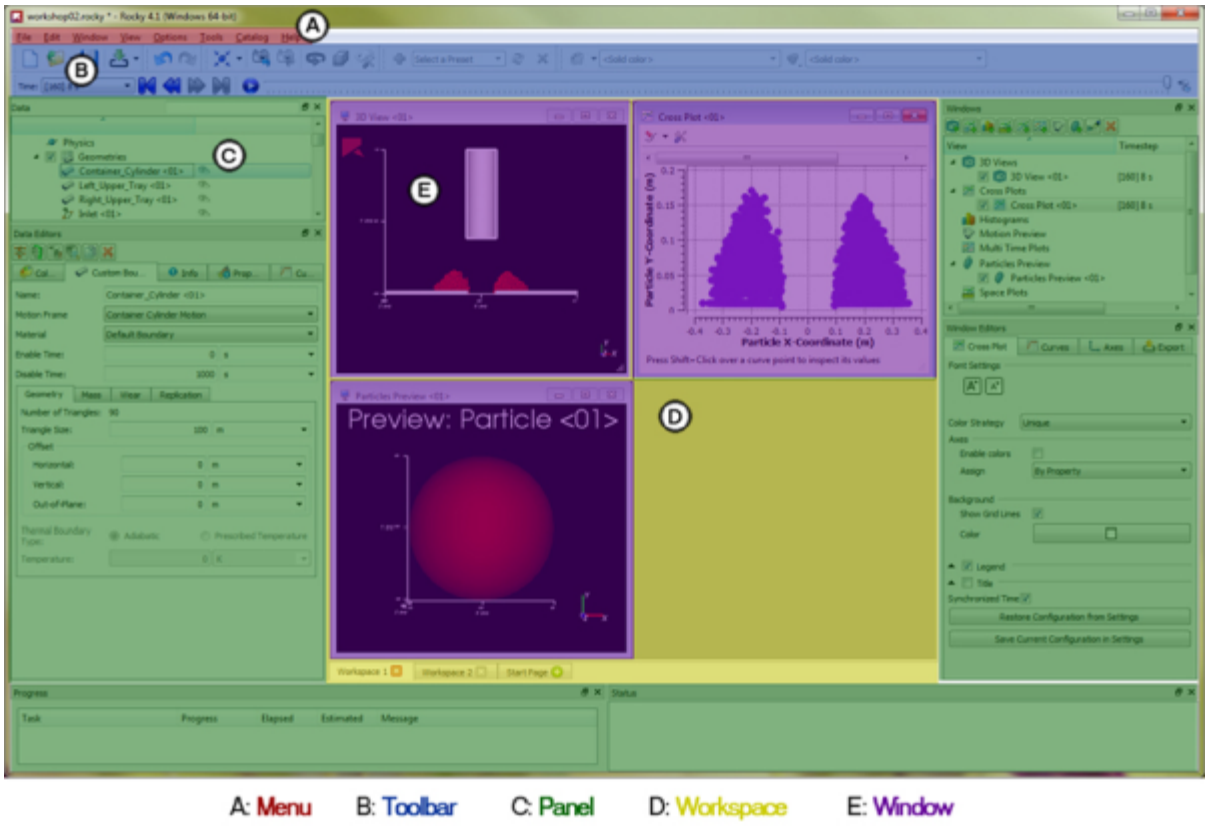
- Getting Started (p. 73)

2.2.1. Main UI Components

Use this section to explore the five major parts of the Rocky user interface, and learn when and why you might access them.

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

Figure 2.2: The five main components of the Rocky user interface



What would you like to do?

- Learn more About the Menu (p. 75)
- Learn more About the Toolbar (p. 77)
- Learn more About Panels (p. 78)
- Learn more About Workspaces (p. 108)

See Also:

- Understand the User Interface (p. 74)

2.2.2. Menu

The menu is always located at the very top of the Rocky main screen and is one of the few UI elements in Rocky that is static in its location. From the Rocky menu, you can do all of the following tasks:

- From the **File** menu:
  - Begin a new simulation project (see also Begin a New Simulation Project (p. 412)).
  - Open an existing simulation project (see also Open an Existing Simulation Project (p. 412)).
  - Save a new simulation project (see also Save a New Simulation Project (p. 853)).

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

  - Archive a simulation project (see also Archive a Simulation Project (p. 1294)).
  - Restore an archived project (see also Restore an Archived Simulation Project (p. 413)).
  - Export or import project context (see also Reuse 3D View Window, Workspace, and User Process Setups in Other Projects (p. 403)).
- From the **Study** menu:
  - Create Regions of Interest.
  - Create Point Cloud.
- From the **Edit** menu:
  - Undo or Redo actions made in Rocky (see also About the Undo/Redo History Panel (p. 106)).
  - Copy an Image or Data Set to the clipboard (see also Copy an Image or Data Set to the Clipboard (p. 1164)).
- From the **Window** menu, you can create, organize (see also Edit or Remove a Workspace (p. 111)), and save an image of (see also Save an Image of the 3D View, Plot, or Histogram (p. 858)) various Rocky windows (p. 1396), including:
  - 3D View windows (see also Create and Modify a 3D View (p. 1040)).
  - Plot and Histogram windows (see also Graphing (Plot or Histogram) a Data Set Within Rocky (p. 1101)).
  - Motion Preview windows (see also Preview a Motion in 3D (p. 573)).
  - Particles Details windows (see also Preview a Particle Shape in 3D (p. 681)).
- From the **View** menu:
  - Show/hide various Rocky panels (p. 1393) (see also About Panels (p. 78)).
  - Revert to the default Rocky user interface layout by clicking **Reset layout**.
  - Change the background color of the Rocky interface by selecting a **Theme**. In this version, there are two options:
    - → **Dark Theme**,(default) which has a mostly black background with light-colored text.
    - → **Light Theme**, which has a mostly white background with dark-colored text.
- From the **Options** menu:
  - Set global preferences for using Rocky (see also About Setting Global Preferences (p. 227)).
  - Set the units used in Rocky (see also About Setting Your Unit System (p. 238)).
  - Install coupling components required for use with Ansys Fluent (see also Install Ansys Coupling Components (p. 792)).

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

- From the **Tools** menu, you can show/hide all of the following panels (p. 1393):
  - The PrePost Script panel (see also About Creating and Using PrePost Scripts (p. 296)).
  - The Python Shell panel (see also Customize and Extend Rocky with Python (p. 294)).
  - The Inspection panel (see also About the Cell Inspector (p. 1211)).
  - The Animation panel (see also About Creating and Saving an Animation (p. 1166)).
  - The Expressions/Variables panel (see also Defining Your Variables (p. 243)).
- From the **Help** menu, you can do all of the following:
  - Review details about this Rocky release by clicking **About**. (See also About This Version of Rocky (p. 50).)
  - Access your Rocky License status and info by clicking **License Info**.
  - Review what changed in this release of Rocky by clicking **Release Notes**.
  - Learn about the key improvements made for this release of Rocky by clicking **What's New**.
  - Access the User Manual (this document), DEM Technical Manual, and other resources by pointing to **Manuals** and then selecting which option you want. (See also Getting Help and Support (p. 1285).)
  - **Tutorial Workshops** where you can access all Rocky Tutorials.
  - **Community Forum** where you can navigate through Rocky community.
  - **Ready-to-use Modules** where you access external modules and their documentation.
  - **Ready-to-use Scripts** where you access external modules and their documentation.
  - Review the technical library, the latest Rocky applications and other resources by pointing to **Website** and then clicking the option you want to access.
  - Access the new versions, Additional packages, External Modules and other features by pointing to **Customer Portal** and then selecting which item you want.
  - **Contact Customer Support** to get support from Rocky team of experts.
  - To be part of **Ansys Product Improvement program**.
  - Ansys Engineering Copilot [BETA] (p. 1286)

See Also:

- Main UI Components (p. 74)

2.2.3. Toolbar

The toolbar is, by default, located at the top of the Rocky main screen just below the menu. There are several individual toolbar components that can make up the main Rocky toolbar, including the following:

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

- Camera visualization toolbar (see also About Using the Camera Visualization Toolbar (p. 1045))
- Coloring service toolbar (see also About Using the Coloring Service Toolbar to Change a 3D View (p. 1084))
- Custom Presets toolbar (see also About the Custom Presets Toolbar (p. 1094))
- File toolbar
- Time toolbar (see also About the Time Toolbar (p. 1279))
- Undo Redo toolbar

These individual toolbar components can be moved or made to float independent of the screen by clicking a toolbar handle and then dragging it and dropping it with the mouse. You can also show/hide individual toolbars by right clicking a toolbar and selecting or clearing the options at the very bottom of the contextual menu.

See Also:

- Main UI Components (p. 74)

2.2.4. Panels

Below the toolbar and surrounding the Workspace are various floating panels that enable you to set parameters and choose options for your simulation and its resulting data. The primary panels for configuring your simulation are the Data and Data Editors panels that appear (by default) to the left of the Workspace; the primary panels for post-processing are the Windows and Windows Editors panels that appear (by default) to the right of the Workspace. Also showing (by default) to the right of the Workspace, you can display various tools panels such as Animations or PrePost Script. You can choose to show/hide multiple panels through both the **View** and **Tools** menus located at the top of the screen.

All panels in Rocky can be resized, moved, nested with other panels, and undocked to float independent of the Rocky main screen. Specifically:

- To resize a panel, hover your mouse over a panel boundary until the resize icon appears. Then, click and drag the boundary to the desired width or height.
- To move a panel to another docked location, click the panel title and drag to the desired area outside of the Workspace. While still dragging the panel, a preview of new docked locations will be displayed with a blue background. To select a new docked location, release the mouse over the preview of the location you want.
- To make a nested panel, click the panel title and then drag it directly over another panel until the lower panel is blue, and then release the mouse. The second panel will appear as a tab at the bottom of the first panel. Select the tab you want to toggle between panels.
- To make a docked panel float, click the panel title, drag it towards the center of the screen then release the mouse. Alternatively, you can also click the icon to the left of the panel's close button to make any docked panel float.

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

Show/hide multiple panels by right clicking a panel's title bar and selecting or clearing the options at the top and middle of the contextual menu.

What would you like to do?

- Learn more About the Data Panel (p. 79)
- Learn more About the Data Editors Panel (p. 82)
- Learn more About the Windows Panel (p. 97)
- Learn more About the Window Editors Panel (p. 98)
- Learn more About the Undo/Redo History Panel (p. 106)
- Learn more About the Log Panel (p. 99)
- Learn more About the Simulation Log Panel (p. 101)
- Learn more About the Cache Panel (p. 103)
- Learn more About the Debug Info Panel (p. 105)
- Learn more About the Progress Panel (p. 1281)
- Learn more About the Status Panel (p. 117)
- Learn more About the PrePost Script Panel (p. 296)
- Learn more About the Inspector Panel (p. 1211)
- Learn more About the Animations Panel (p. 1166)
- Learn more About the Python Shell Panel (p. 294)
- Learn more About the Expressions/Variables Panel (p. 243)

See Also:

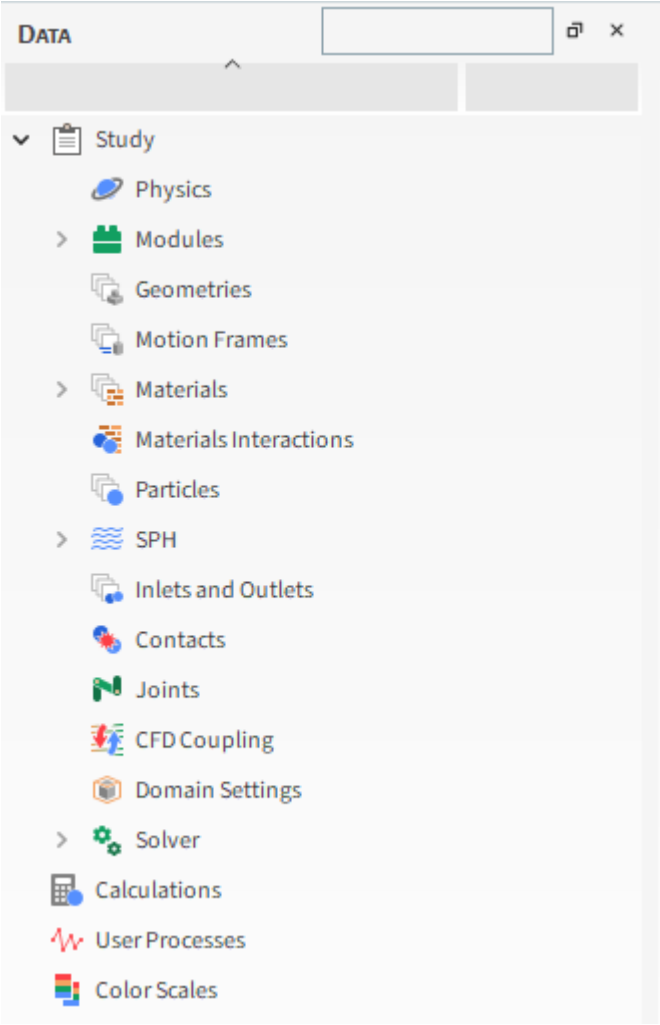
- Main UI Components (p. 74)

2.2.5. Data Panel

The Data panel is the area you use to select the entity with which you want to interact. The tree-like structure is hierarchical and is roughly organized by the steps required to setup, process, and analyze your project. (See Figure 1.)

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

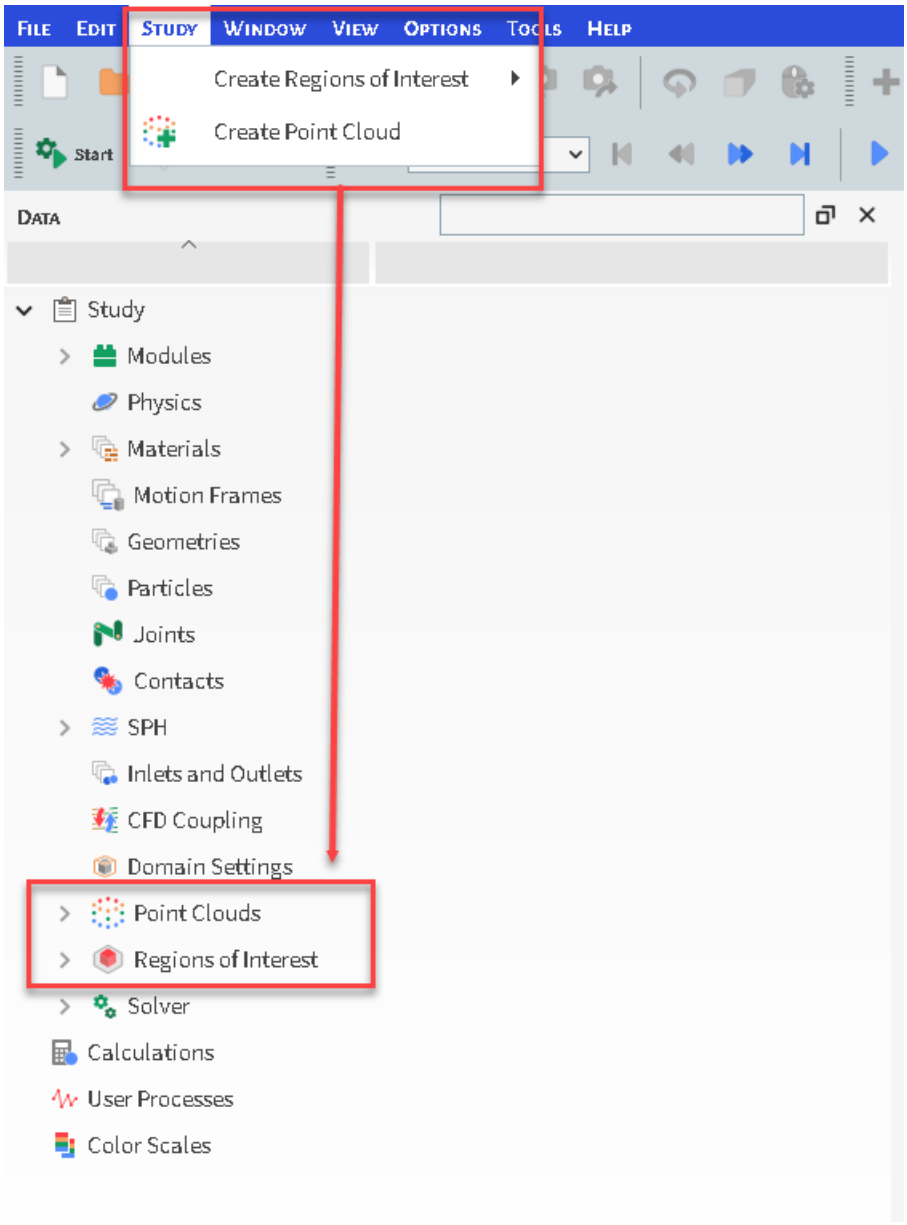
Figure 2.3: Data Panel, default view



By clicking **Study** on the menu, more parameters can be added to the Data Panel, as shown in the image below:

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

Figure 2.4: Data Panel, additional Study parameters



The top-level items in the Data panel include the following entities:

- **Study**, which organizes all the various set-up and processing categories. (See also Set Simulation Parameters (p. 414) and Processing a Simulation (p. 907).)
- **Calculations**, which lists any Particles or SPH elements Properties you have applied to User Processes for which you want to retain calculations after the particle/SPH element leaves the defined area. (See also About Particles Calculations (p. 1242).)
- **User Processes**, which lists the shapes, planes, and settings you have divided or filtered your simulation area into for more focused analysis. (See also Filter Views and Data with User Processes (p. 1174).)
- **Color Scales**, which lists all the Properties you have shown in a 3D View for the purpose of changing the coloration of the legends. (See also Create and Modify a 3D View (p. 1040).)

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

Once selected in the Data panel, options for the item are generally shown in the Data Editors panel (see also About the Data Editors Panel (p. 82)), or through the item's right-click menu. Note: With the exception of the Study item, which has its own settings, top-level Data panel items are for categorization only and do not have settings associated with them. Only the sub-items listed beneath these categories will have settings upon which you can take action.

The Data panel also includes a search box to help you find items in the panel quickly. (See also Use the Data Panel Search Bar to Quickly Find Components (p. 118).) You may also duplicate many of the items in the Data panel to save time on setup. (See also Duplicate a Data Panel Item (p. 846).)

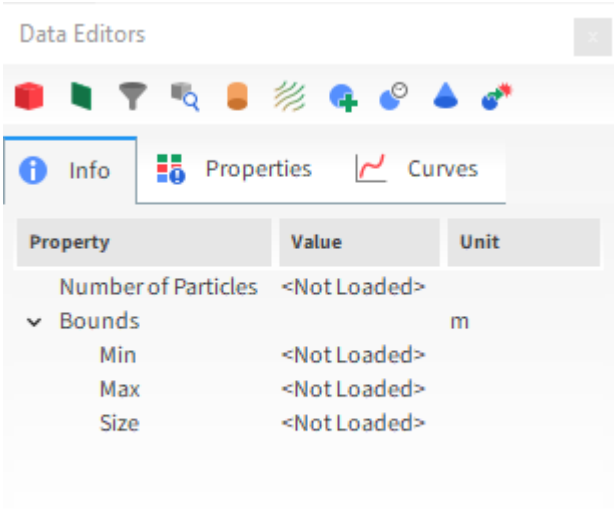
See Also:

- About Panels (p. 78)

2.2.6. Data Editors Panel

The Data Editors panel shows the settings and parameters available for the item selected in the Data panel (see also About the Data Panel (p. 79)). Figure 1 shows the Data Editors panel view when Particles is chosen in the Data panel.

Figure 2.5: Data Editors panel for the Particles entity



What is shown for an entity on the Data Editors panel can include action buttons, like those provided at the top of Figure 1, and settings and information organized into various tabs, such as the following:

- Coloring tab (see also About the Coloring Tab (p. 93))
- Info tab (see also About the Info Tab (p. 83))
- Properties tab (see also About Properties (p. 933))
- Curves tab (see also About Curves (p. 989))

Some entities in the Data panel, such as Particles Calculations and Color Scales, are categories only and will have no information displayed in the Data Editors panel. However, sub-items within these categories will have information upon which you can take action.

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

See Also:

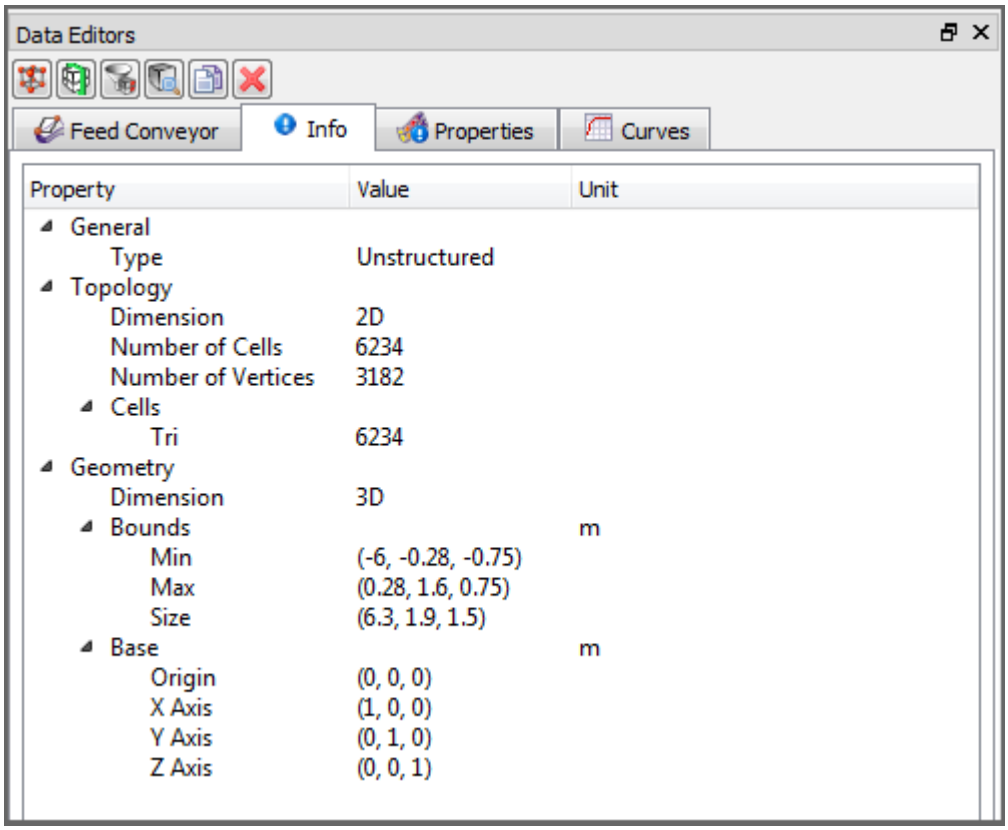
- About Panels (p. 78)

2.2.7. Info Tab

The Info tab is one of several tabs that appear on the **Data Editors** panel when a simulation entity (p. 1395) is selected in the **Data** panel. Though purely informational, the Info tab does show some data about the selected entity that could be useful when setting up or analyzing your simulation.

For example, the Info tab can display domain properties, such as the topology and geometry coordinate limits, that provide context as to how the Property (p. 1394) simulation data was recorded. These properties are displayed on the Info tab for simulation entities including conveyors (Figure 1) and imported geometries, the main Particles entity, 1-Way LBM, 1-Way Fluent, and individual User Processes.

Figure 2.6: Info tab showing results for a Feed Conveyor

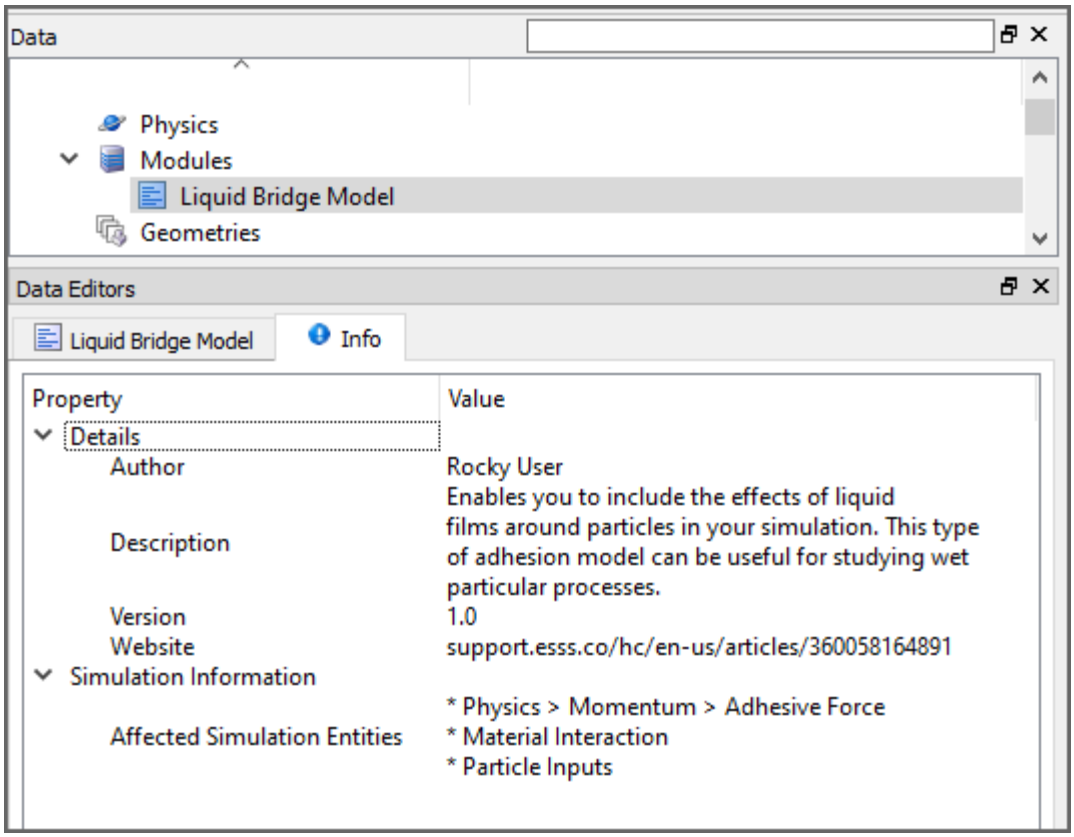


ABOUT THE INFO TAB FOR MODULES

The Info tab on individual Modules (see also About Modules Parameters (p. 431)) describes the Author and Version details for the Module, and lists what other entities in the Rocky UI are affected by enabling that particular Module. For example, the Info tab for the Liquid Bridge Model Module (Figure 2) describes three places in the Rocky UI affected by that module.

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

Figure 2.7: Info tab for the Liquid Bridge Model Module



Note: In this version of Rocky, the **Liquid Bridge Model** Module is provided as an external (p. 1391) module. Refer to the Install an External Module (p. 223) topic for details.

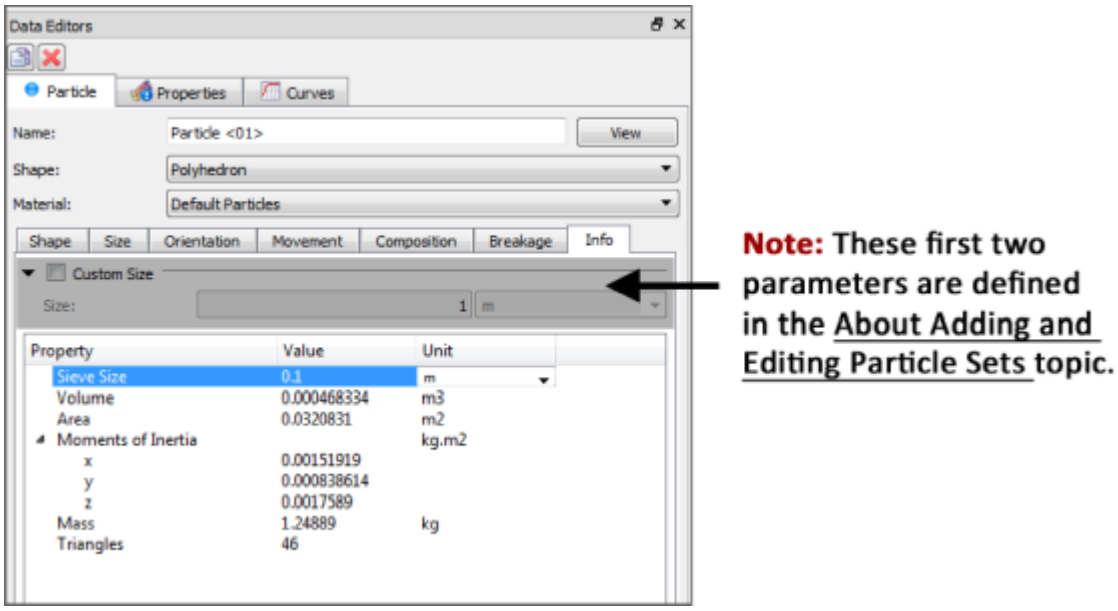
This information can help you verify that the parameters in those locations are set correctly for the feature enabled by the module.

ABOUT THE INFO TAB FOR INDIVIDUAL PARTICLE SETS

When an individual Particle set is selected under the main Particles entity, the Info tab shows useful information about the largest particle in the selected particle set, but also contains additional functionality unique to that tab (Figure 3).

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

Figure 2.8: Info tab showing results for an individual Particle set



For example, you can change the units by clicking a unit and selecting a new type from the list that appears. You can also use the **Custom Size** (**Custom Diameter** or **Custom Scale Factor**) checkbox to experiment with different particle sizes and how they affect the properties displayed on the Info tab without having to switch back to the **Size** tab. Note: This functionality is explained in more detail in the About Adding and Editing Particle Sets (p. 602) topic.

ABOUT THE MAX ADMISSIBLE OVERLAP VALUE(for Custom Concave Particles)

The **Max Admissible Overlap** value is provided only for custom (imported) concave particles. Its purpose is to display the maximum percentage of concave overlap that will be allowed during the simulation.

As a comparison, for a sphere and a boundary collision, the maximum overlap allowed is 50%— above that the sphere center would be on the other side of the boundary and the particle would cross it. For concave particles, the maximum admissible overlap depends on the shape. Ideally, the maximum admissible overlap should not be too low, or you will have to compensate for that by using a larger Young's modulus for stability.

Tip: If you have imported a concave particle with values too small for the maximum overlap allowed, consider modifying the STL shape to result in higher values. This might help you avoid numerical issues.

Use the following information to help you better understand this value and how best to use it:

- **What is this value used for?** It is a reference that can help you prevent excessive overlaps that can cause numerical instabilities for the concave particle shape used in the simulation. The smaller the maximum admissible overlap, the smaller are the overlap values that are tolerated during the simulation.
- **How is this value calculated?** It is computed based upon the minimum distance from the particle's triangle centers to other triangles of the particle in the direction perpendicular to the triangle's plane. So the max admissible overlap is a function of the particle shape. Sharp edges will result in lower maximum admissible overlaps to guarantee the solver's stability.

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

- **How does this value relate to the STL file I imported?** This is not a quality criteria for the STL file itself; rather, it is related to the particle shape. If the shape has more rounded or sharp edges, it will be reflected on the max admissible overlap value. So more (or fewer) overlaps will be admitted to ensure the stability of the simulations.

ABOUT THE INFO TAB FOR INPUT ENTITIES

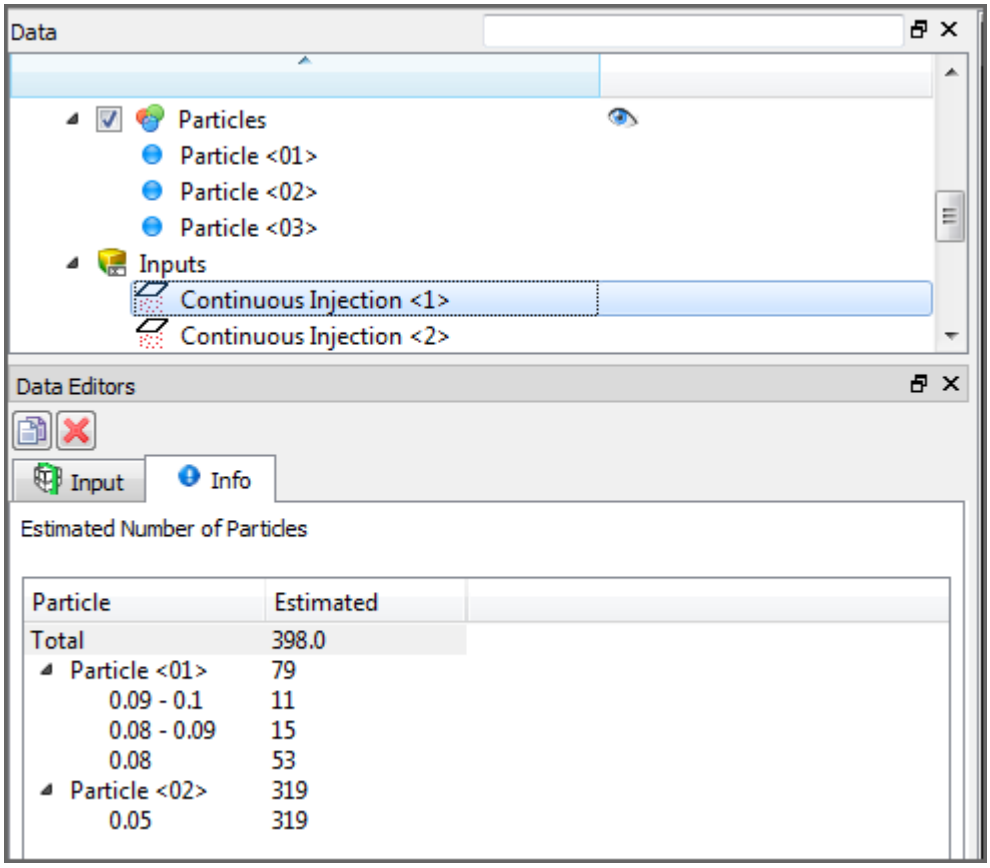
For Inputs (Figure 4), the Info tab displays the simulation particle count in different ways depending upon the particle size distribution that was specified:

- For Particle sets made up of only one size (no size distribution), the Info tab shows the exact number of particles that will be released into the simulation.
- For Particle sets with a size distribution, the Info tab provides an estimate of the number of particles that could be released into the simulation.

Note:

For Volume Fill specifically, the estimate assumes there are no boundary limits to consider, and assumes a **Gap Scale Factor** equal to **1**. This means that if you have set your particle Mass higher than the box and gap you define can contain, the particle estimate on the Info tab will show a higher number than what will actually be injected.

Figure 2.9: Info tab showing results for an individual Particle Input



Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

For Particle sets with size ranges specified, viewing the **Simulation Summary** after processing has begun is still the only place to get an exact particle count; however, if a particle count estimate is sufficient, the particle count on the Input entity's Info tab can be viewed at any time during project setup. (See also About the Simulation Summary (p. 859).)

INFO TAB PROPERTY DEFINITIONS

See tables below to help you understand more about the Info tab.

Table 1: Info tab definitions for individual Geometry components, Inlets, the main Particles entity, 1-Way LBM, 1-Way Fluent, and individual User Processes (Figure 1 above)

Property	Description	Values Displayed
General		
Type	Indicates the type of grid the domain property is based upon. Specifically: • **Unstructured** is for Particles and indicates that each *node* location will be unrestricted by any sort of grid. • **Structured** is for geometry components and indicates that each *triangle* location will be restricted by a grid whose dimensions are defined by the smallest triangle size.	Unstructured; Structured
Topology		
Dimension	Indicates whether the domain property has no dimension; or is one dimensional (*node*), two dimensional (*triangle*), or three dimensional (*block* or *cell*).	0D; 1D; 2D; 3D
Number of Cells	Lists the number of *cell* within the simulation bounds.	Positive whole numbers
Number of Vertices	The number of cell intersections within the simulation bounds. For geometries, this is the points between *triangle*. For Eulerian Statistics, this is the corner points making up each *block* And so on.	Positive whole numbers
Cells Point	For Particles, lists the number of points within the simulation bounds.	Positive whole numbers
Cells Tri	For Geometry components, lists the number of triangles within the simulation bounds.	Positive whole numbers
Cells Bar	For Particle Trajectory calculations, lists the total number of trajectory bars that were calculated.	Positive whole numbers
Shape	For Eulerian Statistics, indicates the number of divisions that were defined in the X, Y, and Z directions respectively.	Whole values greater than or equal to 1
Geometry		

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

Property	Description	Values Displayed
Dimension	Indicates whether the geometry components within the simulation have no dimension; or are one dimensional (*node*), two dimensional (*triangle*), or three dimensional (*block* or *cell*).	0D; 1D; 2D; 3D
Bounds Min	The minimum coordinates of the simulation data.	X, Y, Z
Bounds Max	The maximum coordinates of the simulation data.	X, Y, Z
Bounds Size	The size dimensions of the current simulation bounds.	X, Y, Z
Length	For Particle Trajectory calculations, provides the total length of the trajectory lines added together.	Positive values
Volume	For Eulerian Statistics, provides the total volume of all the Eulerian blocks added together.	Positive values

Table 2: Info tab definitions for individual Modules (Figure 2 above) (See also About Modules Parameters (p. 431).)

Property	Description	Values Displayed
Details		
Author	If provided, lists the name of the developer or company that designed and built the Module.	Text
Description	If provided, displays a short description of what the Module does.	Text
Version	If provided, displays the Module version. For Modules created by Ansys, this will be equal to the version of Rocky for which the Module was designed.	Text
Website	If provided, displays the URL of the website associated with the Author of the Module.	Text
Simulation Information		
Affected Simulation Entities	Describes whether or not enabling the Module caused changes or additional settings to be made in other areas of the Rocky UI. Specifically: • If the Module affects no other areas of the Rocky UI, then the message will say "Module does not affect any simulation entity." • If the Module does affect other areas of the Rocky UI, those areas will be listed.	Text

Table 3: Info tab definitions for individual Particle sets (Figure 3 above) (See also About Adding and Editing Particle Sets (p. 602).)

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

Property	Description	Values Displayed
Scale Factor	By default, when the **Custom Scale Factor** checkbox is cleared, this displays the size of the largest particle in the *Particle set* using the **Original Size Scale** method of measurement that was selected on the **Size** sub-tab. When the **Custom Scale Factor** checkbox is enabled, this displays the experimental size entered in the corresponding **Scale Factor**) box. (See also About Adding and Editing Particle Sets.)	Positive values
Equivalent Diameter	By default, when the **Custom Diameter** checkbox is cleared, this displays the size of the largest particle in the *Particle set* using the **Equivalent Sphere Diameter** method of measurement that is selected on the **Size** sub-tab. When the **Custom Diameter** checkbox is enabled, this displays the experimental size entered in the corresponding **Diameter** box. (See also About Adding and Editing Particle Sets.)	Positive values
Sieve size	By default, when the **Custom Size** (**Custom Diameter** or **Custom Scale Factor**) checkbox is cleared, no matter what is selected for **Size Type** on the **Size** sub-tab, this converts the size of the largest particle to its Sieve Size value, which is based upon the dimensions of a square hole just big enough for the particle to pass through. When the **Custom Size** (**Custom Diameter** or **Custom Scale Factor**) checkbox is enabled, this displays the experimental size entered in the corresponding **Size** (**Diameter** or **Scale Factor**) box converted into its Sieve Size value. (See also About Adding and Editing Particle Sets.)	Positive values
Volume	By default, when the **Custom Size** (**Custom Diameter** or **Custom Scale Factor**) checkbox is cleared, this displays the volume of the largest particle in the Particle set. When the **Custom Size** (**Custom Diameter** or **Custom Scale Factor**) checkbox is enabled, this displays the volume of the experimental particle defined in the corresponding **Size** (**Diameter** or **Scale Factor**) box. (See also About Adding and Editing Particle Sets.)	Positive values
Area	By default, when the **Custom Size** (**Custom Diameter** or **Custom Scale Factor**) checkbox is cleared, this displays the surface area of the largest particle in the Particle set. When the **Custom Size** (**Custom Diameter** or **Custom Scale Factor**) checkbox is enabled, this displays the surface area of the experimental particle defined in the corresponding **Size** (**Diameter** or **Scale Factor**) box. (See also About Adding and Editing Particle Sets.)	Positive values
Moments of Inertia x	By default, when the **Custom Size** (**Custom Diameter** or **Custom Scale Factor**) checkbox is cleared, this displays the moment of inertia in the X direction of the largest particle in the Particle set. When the **Custom Size** (**Custom Diameter** or **Custom Scale Factor**) checkbox is enabled, this displays the moment of inertia in the X direction for the experimental particle defined in the corresponding **Size** (**Diameter** or **Scale Factor**) box. (See also About Adding and Editing Particle Sets.)	Positive values
Moments of Inertia y	By default, when the **Custom Size** (**Custom Diameter** or **Custom Scale Factor**) checkbox is cleared, this displays the moment of inertia in the Y direction of the largest particle in the Particle set. When the **Custom Size** (**Custom Diameter** or **Custom Scale Factor**) checkbox is enabled, this displays the moment of inertia in the Y direction for the experimental	Positive values

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

Property	Description	Values Displayed
	particle defined in the corresponding **Size** (**Diameter** or **Scale Factor**) box. (See also About Adding and Editing Particle Sets.)	
Moments of Inertia z	By default, when the **Custom Size** (**Custom Diameter** or **Custom Scale Factor**) checkbox is cleared, this displays the moment of inertia in the Z direction of the largest particle in the Particle set. When the **Custom Size** (**Custom Diameter** or **Custom Scale Factor**) checkbox is enabled, this displays the moment of inertia in the Z direction for the experimental particle defined in the corresponding **Size** (**Diameter** or **Scale Factor**) box. (See also About Adding and Editing Particle Sets.)	Positive values
Mass	By default, when the **Custom Size** (**Custom Diameter** or **Custom Scale Factor**) checkbox is cleared, this displays the mass of the largest particle in the Particle set. When the **Custom Size** (**Custom Diameter** or **Custom Scale Factor**) checkbox is enabled, this displays the mass of the experimental particle defined in the corresponding **Size** (**Diameter** or **Scale Factor**) box. (See also About Adding and Editing Particle Sets.)	Positive values
Elements	For Particle sets with **Multiple Elements** selected on the **Composition** sub-tab, this displays the number of individual *element* that the particle shape will be divided into. Specifically: • By default, when the **Custom Size** (**Custom Diameter** or **Custom Scale Factor**) checkbox is cleared, this value is based on the size specified on the **Size** sub-tab. • When the **Custom Size** (**Custom Diameter** or **Custom Scale Factor**) checkbox is enabled, this value is based upon the experimental particle defined in the corresponding **Size** (**Diameter** or **Scale Factor**) box. (See also About Adding and Editing Particle Sets.)	Positive values
Triangles	By default, when the **Custom Size** (**Custom Diameter** or **Custom Scale Factor**) checkbox is cleared, this displays the number of triangles that make up the largest particle in the Particle set. When the **Custom Size** (**Custom Diameter** or **Custom Scale Factor**) checkbox is enabled, this displays the number of triangles making up the experimental particle defined in the corresponding **Size** (**Diameter** or **Scale Factor**) box. (See also About Adding and Editing Particle Sets.) Note: The way in which particle shape triangles are counted in Rocky depend upon the particle shape and composition. Specifically: • For *Solid* particles composed of multiple elements (also known as *meshed*), this is the number of triangles in the outer surface of the particle, and does not include the triangles shared between adjacent *element*. • For *Shell* particles composed of multiple elements, this is the number of individual elements where each element is represented as a triangle with a defined thickness.	Positive values

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

Property	Description	Values Displayed
	• For Shell particles with single-element compositions (non-meshed), this is the number of triangles present in the original imported .stl shape file. • For Polyhedrons, Custom Polyhedrons and Faceted Cylinders composed of single elements, this value is equal to the number of triangles shown on the Particles Details window. • For all other particle shapes-which include Spheres, Sphero-Cylinders, Briquettes, Straight Fibers, and Custom Fibers-Rocky does not use triangles for these particle shapes during the simulation and therefore **Triangles** will not display on the Summary. Note: For these shapes, the triangles seen in the Particles Details window are for visualization purposes only.	
Vertices	For Particle sets made up of custom (imported) particle shapes, this displays the number of points between the *triangle* (for Custom Shells or Custom Polyhedrons) or between the *segments* (for Custom Fibers) that make up the particle shape. Specifically: • By default, when the **Custom Size** (**Custom Diameter** or **Custom Scale Factor**) checkbox is cleared, this value is based on the largest particle size that is specified on the **Size** sub-tab. • When the **Custom Size** (**Custom Diameter** or **Custom Scale Factor**) checkbox is enabled, this value is based upon the experimental particle defined in the corresponding **Size** (**Diameter** or **Scale Factor**) box. (See also About Adding and Editing Particle Sets.) Note: The way in which vertices are counted in Rocky depend upon the particle shape that was imported. Specifically: • For Custom Fibers, this is the number of individual points (joints) at the ends of the segments that compose the fiber. • For Custom Shells and Custom Polyhedrons, this is the number of vertices that compose the triangles in the original .stl shape file.	Positive whole numbers
Max Admissible Overlap	For custom imported concave particles, displays the maximum percentage of concave overlap that will be allowed during the simulation.	0-100
Custom Shape Type	For custom shapes that are imported into Rocky from an .stl file, this displays the category to which Rocky has assigned the shape. Rocky categorizes shapes based upon various attributes and also uses those	Concave; Convex; Fiber; Not

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

Property	Description	Values Displayed
	categories to determine what kind calculations to perform during processing. **Tip:** For more information about how Rocky categorizes custom particle shapes, refer to the DEM Technical Manual. (From the Rocky **Help** menu, point to **Manuals**, and then click **DEM Technical Manual**.)	Custom; Shell

Table 4: Info tab definitions for individual Particle Inputs (Figure 4 above) (See also About Adding and Editing Particle Inputs (p. 708).)

Property	Description	Values Displayed
Total	For the main **Inputs** entity, this displays the total amount of particles that are estimated to be released into the simulation from all individual Particle Inputs. For individual Particle Inputs (Figure 3), this displays the total amount of particles that are estimated to be released into the simulation from only that input. **Note:** For Particle sets with only one size specified, the particle count is exact. For Particle sets with a size distribution, the particle count is an estimate only.	Positive whole numbers
<Particle Input Name> (automatically provided)	For the main **Inputs** entity, this displays the total amount of particles that are estimated to be released into the simulation from only the selected input. **Note:** For Particle sets with only one size specified, the particle count is exact. For Particle sets with a size distribution, the particle count is an estimate only.	Positive whole numbers

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

Property	Description	Values Displayed
<Particle Name> (automatically provided)	For each individual Particle Input listed on either the main **Inputs** entity or on its own Info tab (Figure 3), this displays the amount of particles from only this *Particle set* that are estimated to be released into the simulation from the input to which it is assigned. When expanded, the top most total amount is further broken down by the particle size ranges specified for the Particle set. **Note:** For Particle sets with only one size specified, the particle count is exact. For Particle sets with a size distribution, the particle count is an estimate only.	Positive whole numbers

Tip:

If you see **<Not Loaded>** listed for items under the **Value** column, right-click the item and then click **Load…** to have Rocky calculate and display the values.

See Also:

- About Properties (p. 933)
- About the Data Editors Panel (p. 82)

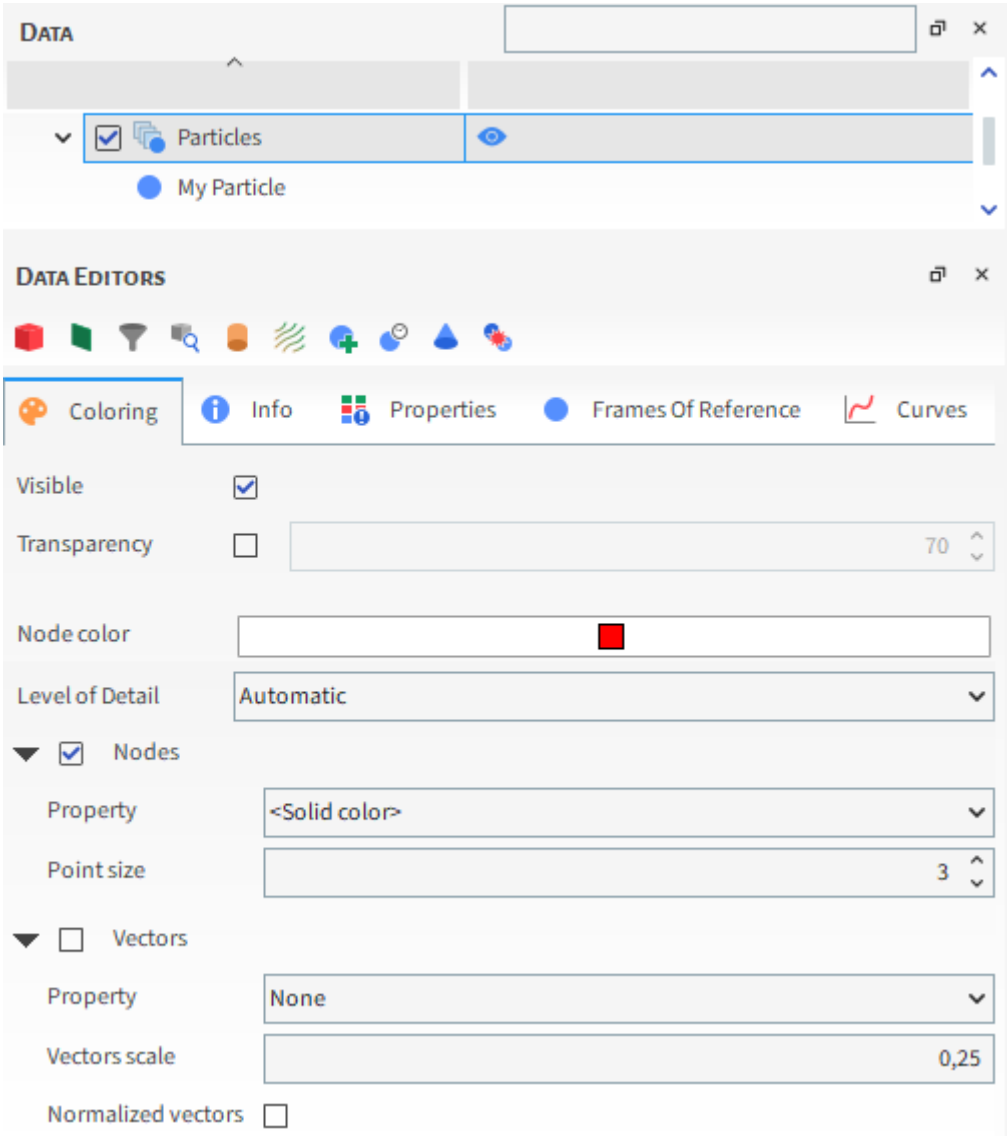
2.2.8. About the Coloring Tab

The Coloring tab, located on the **Data Editors** panel when a simulation entity (p. 1395) is selected in the **Data** panel, helps you change the colors and data attributes of your selected entity.

The options displayed on the tab are dependent upon which type of window you select. If a 3D Plot window is selected, the Coloring tab shows options that enable you to change the appearance of the entity you have selected in the **Data** panel (Figure 1). Those changes will be visible in the 3D View.

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

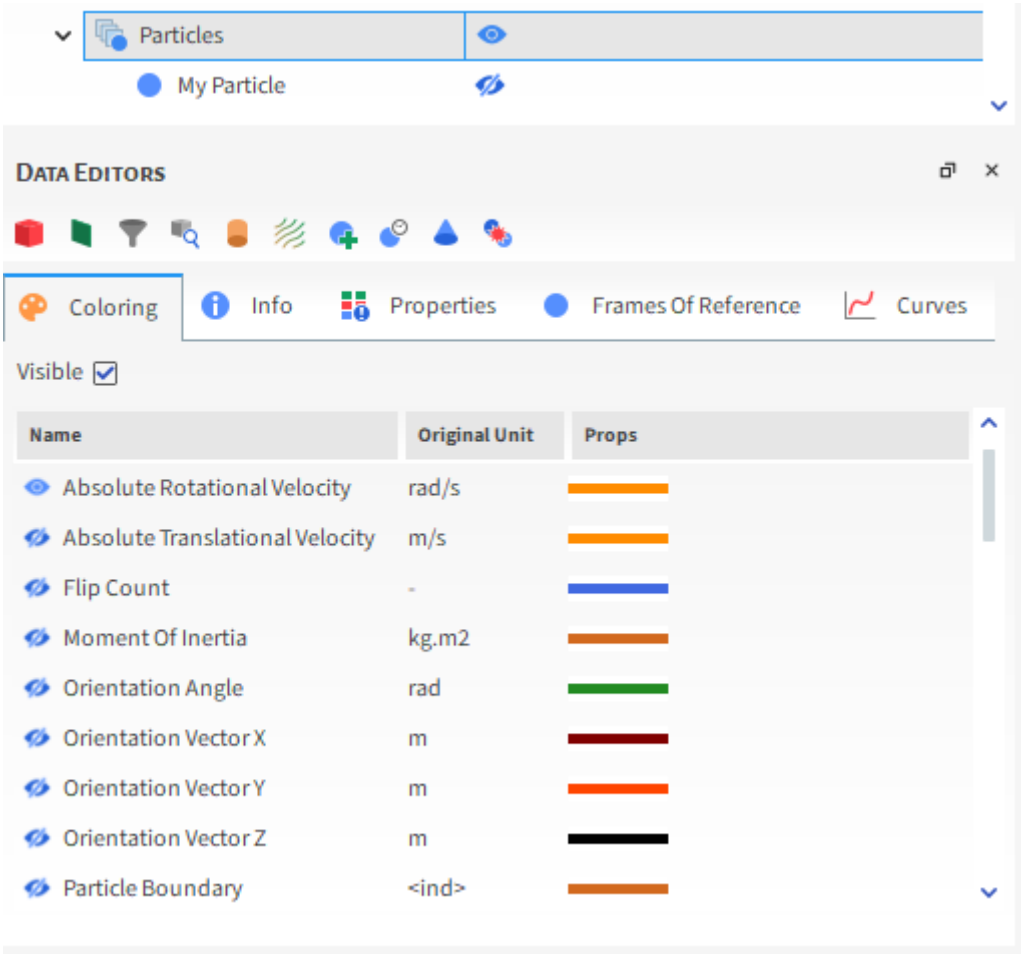
Figure 2.10: Coloring tab for Particles when a 3D View window is selected



If a plot or histogram window is selected, the Coloring tab shows the data sets that are already calculated and ready to display in the window (Figure 2). The active data sets are indicated with an "open" eye icon. (See also Show/Hide Components by Using Eye Icons and Checkboxes (p. 113).) Unlike when a 3D View window is selected, the data on the Coloring tab when a plot or histogram window is selected is not dependent upon what is selected in the **Data** panel.

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

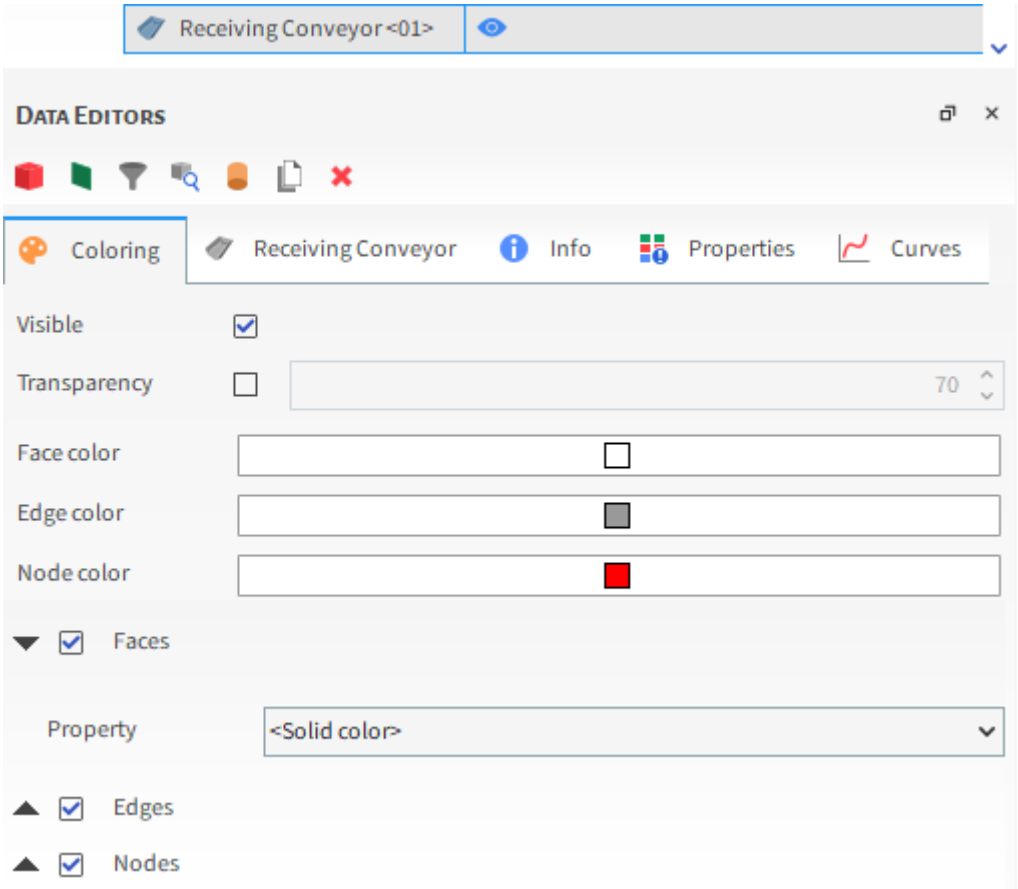
Figure 2.11: Coloring tab when a plot or histogram window is selected



If a Motion Preview window is selected, the Coloring tab shows options related to the axes of the Frame that is selected in the **Data** panel.

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

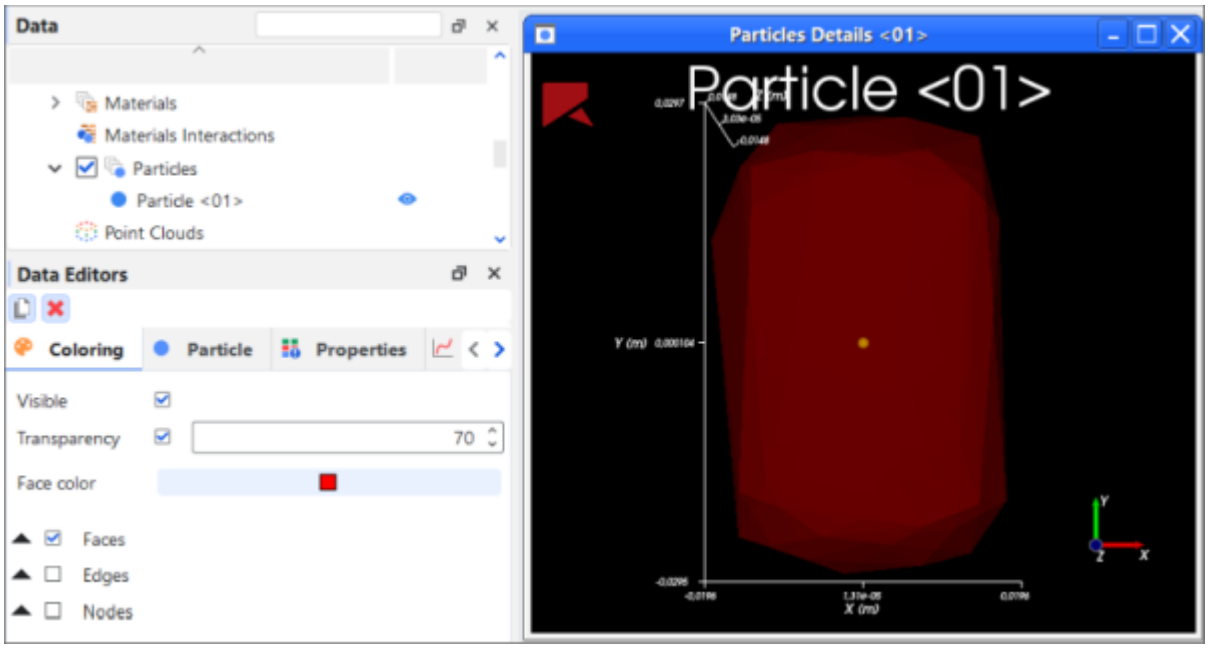
Figure 2.12: Coloring tab when a Motion Frame is selected in the Data panel AND a Motion Preview window is selected



If a Particles Details window is selected, the Coloring tab shows options that will enable you to change the appearance of the particle shape you have selected to display. You can also adjust the transparency settings, allowing you to finely tune the opacity of the particle and achive the desired level of transparency (Figure 4).

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

Figure 2.13: Coloring tab when a Particles Details window is selected



See Also:

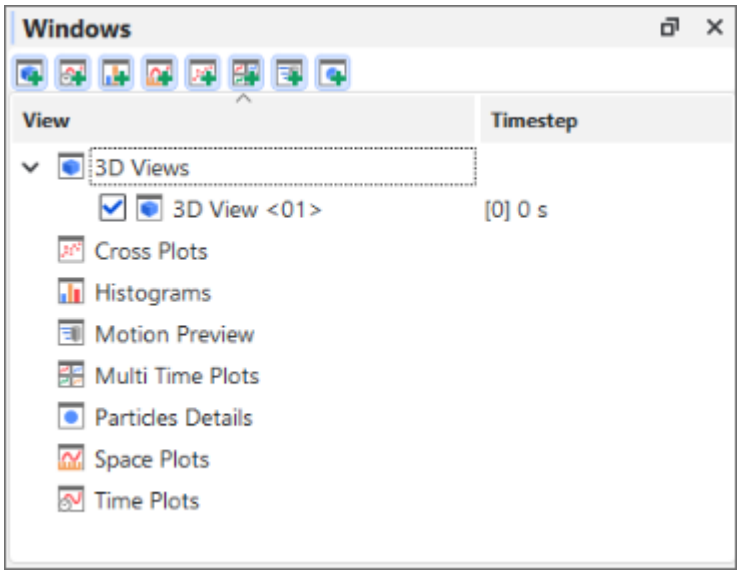
- About Using the Coloring Tab to Change a 3D View (p. 1074)
- Using the Coloring Tab to Change a 3D View (p. 1092)
- Use the Coloring Tab to Change a Plot or Histogram (p. 1154)
- Use the Coloring Tab to Change the Motion Preview for a Frame (p. 582)
- Use the Coloring Tab to Change the Preview for a Particle Set (p. 697)
- About the Data Editors Panel (p. 82)

2.2.9. Windows Panel

The Windows panel enables you to easily view, create, or remove the various Rocky windows-including 3D View windows, Motion Preview windows, and plots and histogram windows-all in one place (Figure 1).

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

Figure 2.14: Windows panel showing one open 3D View window



It is hidden by default when you first open Rocky, but can be shown by selecting **Windows** from the **View** menu.

On the Windows panel, existing windows are listed below their categories, and new windows can be created from the button bar at the top of the panel. And by using the checkboxes, windows can be shown or hidden from the workspace (p. 1396) without actually removing them from the project. (See also Show/Hide Components by Using Eye icons and Checkboxes (p. 113).)

Selecting a window from this panel will enable that window's editable properties in the **Window Editors** panel.

See Also:

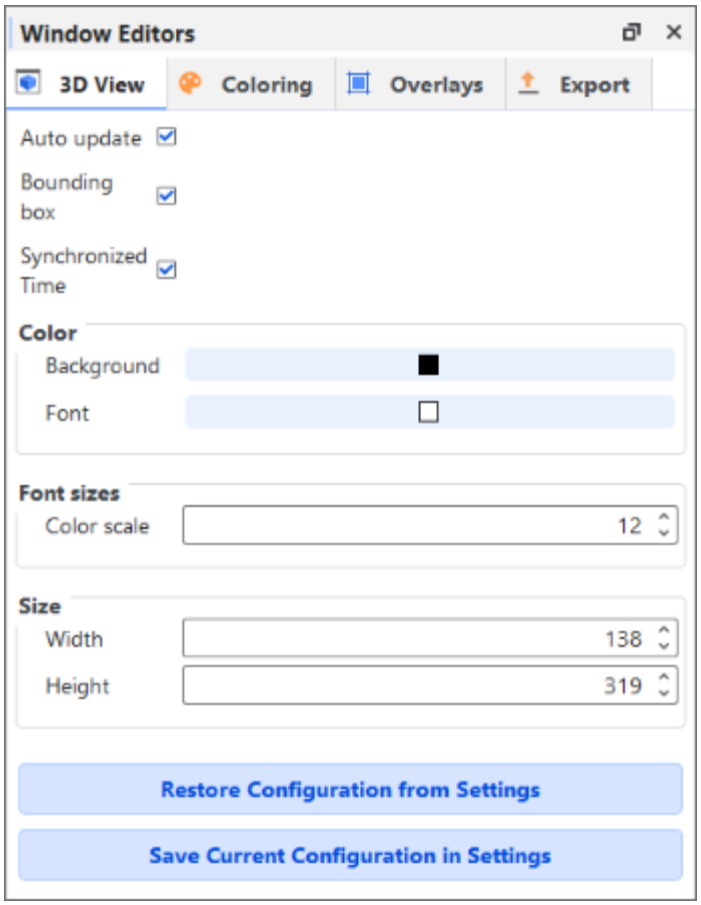
- About Panels (p. 78)
- About the Window Editors Panel (p. 98)

2.2.10. Window Editors Panel

The Window Editors panel (Figure 1) displays the editable settings and parameters for the Rocky window that is selected in the **Windows** panel (see also About the Windows Panel (p. 97)).

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

Figure 2.15: The Window Editors panel showing editable options for a 3D View window



Depending upon the type of window that is selected, different options will be presented, such as the following:

- Window-type-specific options, such as background color, and font sizes.
- Coloring, which provides options for displaying the data (geometries, graph lines, etc.) shown in the selected window.
- Overlays, which provides options for adding text or images to the 3D View window.
- Export, which provides options for exporting and then viewing images or data from the window outside of Rocky.

See Also:

- About Panels (p. 78)

2.2.11. Log Panel

The Log panel is for more advanced users of Rocky. Its purpose is to display the actions you have taken in the program that Rocky records for bug reporting purposes (Figure 1). When an error is shown in the program, you are sometimes given the option of submitting a report with your feedback. That report would contain details regarding the actions displayed in this panel.

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

This kind of information can be useful when writing and testing scripts (p. 1394). (See also About Creating and Using PrePost Scripts (p. 296).)

For example, the actions displayed on the Log panel can be referenced while playing a script (Figure 1), which can be useful for debugging purposes.

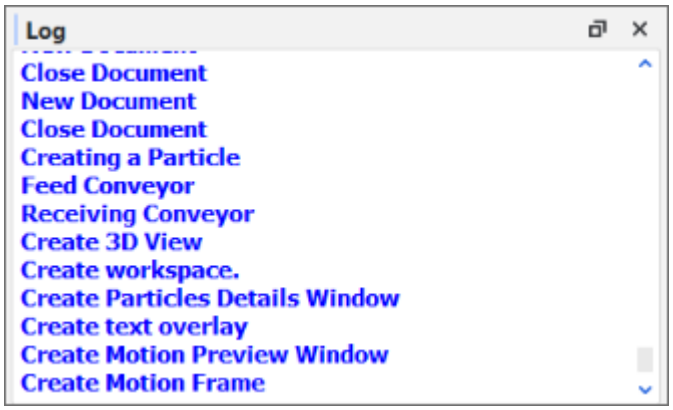


Figure 1: Log panel showing some actions that were recorded

In addition, if a print function is referenced when creating a script (Figure 2), it will appear in the Log panel when the script is played (Figure 3).

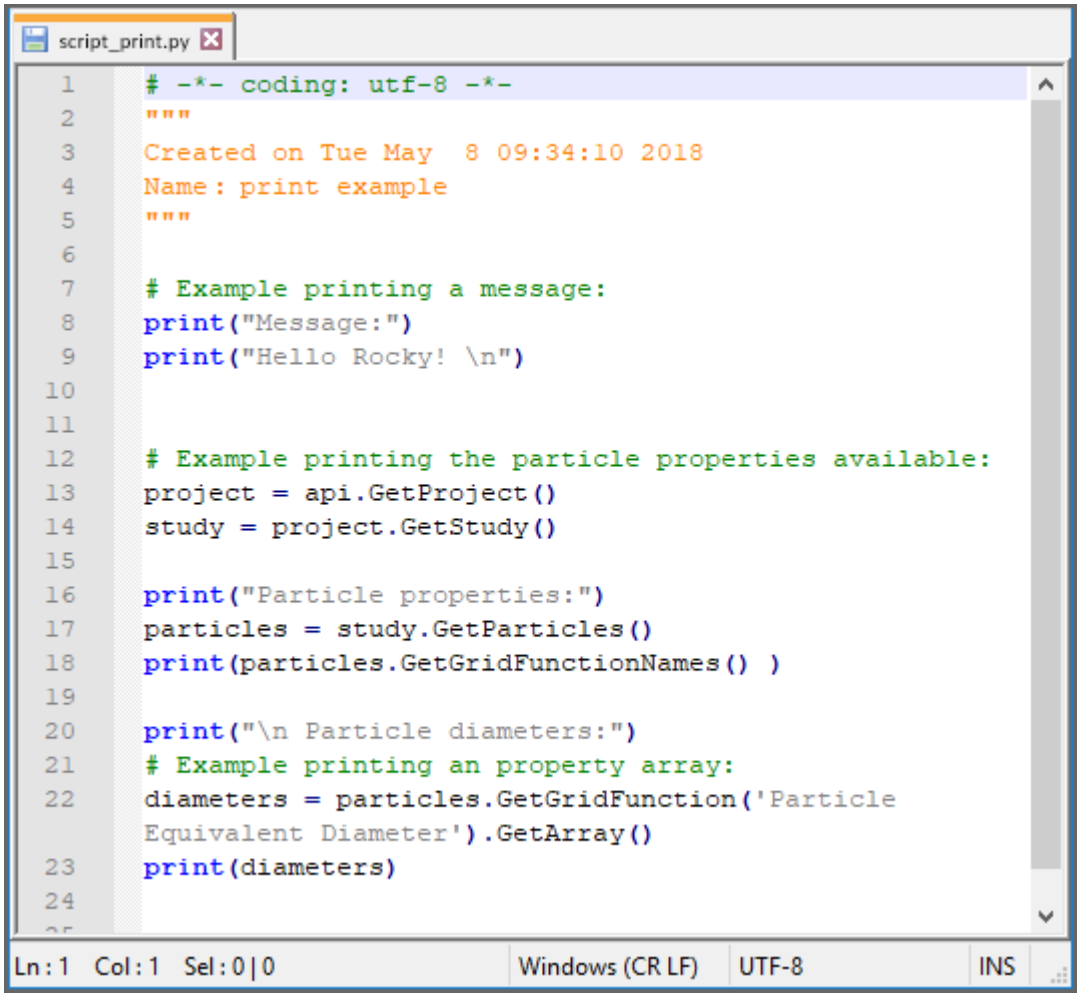


Figure 2: Example script making use of a print function

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

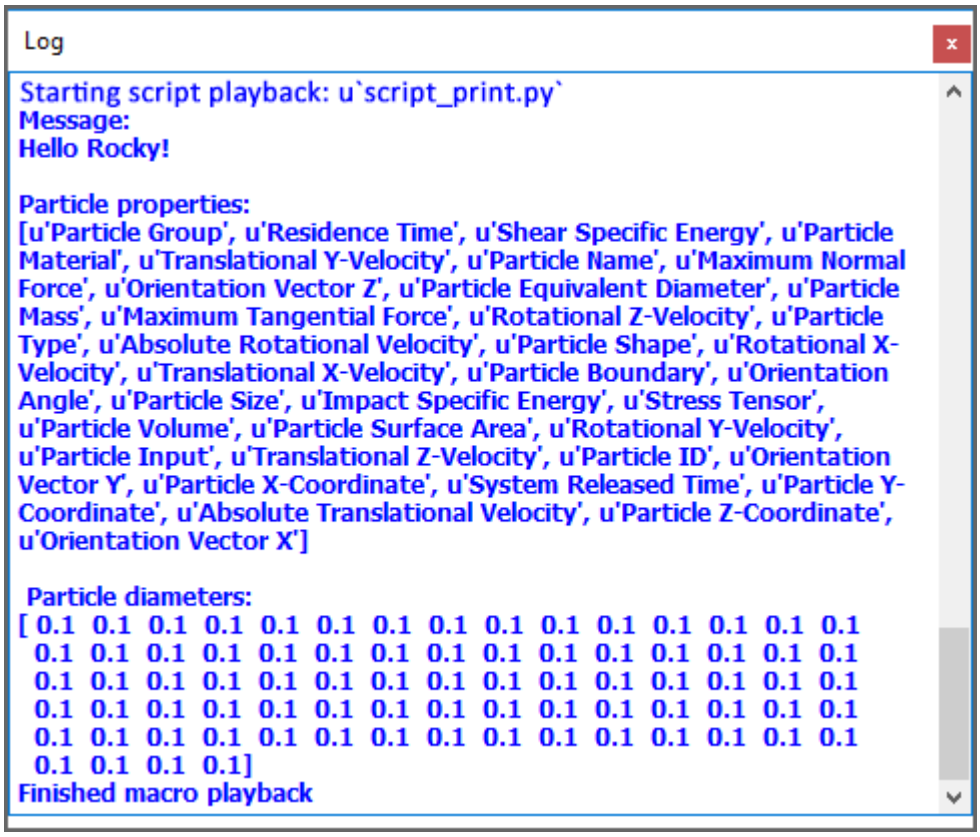


Figure 3: Log panel showing print function when example script is played

The Log is for display only and cannot be edited. Like the Undo/Redo history panel (see also About the Undo/Redo History Panel (p. 106)), the items displayed in this panel are cleared every time you close and open the project.

See Also:

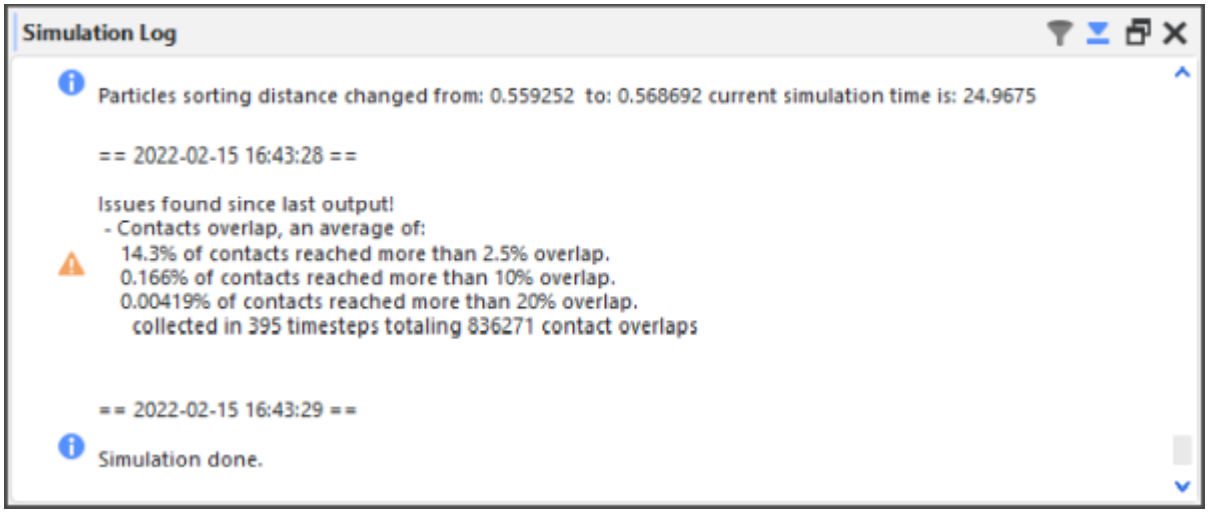
- About Panels (p. 78)

2.2.12. About the Simulation Log Panel

The Simulation Log panel displays any solver-related warnings, errors, or other information that happen during processing (Figure 1), no matter whether you are running your simulations using CPU or GPU resources. This kind of information can be useful when troubleshooting issues that occur when you process your simulation project.

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

Figure 2.16: Simulation Log panel showing some warnings and information that was logged during processing



The Simulation Log is for display only and cannot be edited. But because it is possible to get many messages per simulation, you are able to filter the results and/or have Rocky automatically scroll to the bottom of the list by using the **Filter** and **Auto Scroll** buttons, respectively, at the top of the panel.

Unlike the other Log panels in Rocky, the items displayed in this panel are retained when the project is closed. This is to help you troubleshoot any issues with the Solver that might have caused Rocky to stop processing. It can also be useful in cases where you have used the Rocky Scheduler (see also About Processing Multiple Simulations In Succession (p. 920)) to process a project and want to review what errors or warnings were recorded by the solver during processing.

The Simulation Log is cleared whenever a new simulation or project is started.

Tip: For details about some of the more asked-about warnings and errors that can appear on your Simulation Log panel, see also I get warnings or errors on my Simulation Log Panel (p. 1346).

SIMULATION LOG PANEL FUNCTIONALITY

See the table below for information about the Simulation Log panel functionality.

Table 1: Simulation Log panel functionality

Button	Description	Range
 Filter	Enables you to reduce the information shown to only messages of certain types. Messages are filtered by level of importance. Specifically: • Selecting **Error** shows only error messages. • Selecting **Warning** shows both error and warning messages, but not general information messages.	Information; Warning; Error

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

Button	Description	Range
	• Selecting **Information** shows *all* messages, including general information, errors and warnings. **Tip:** To see all messages of any type, ensure that the Filter is set to **Information**.	
 Auto Scroll	Enables you to determine whether Rocky will automatically scroll the view area in the panel when new information is added, or will keep the view area stationary. **Tip:** To always see the latest messages without having to scroll down in the panel, ensure Auto Scroll is turned on.	Turns on or off

See Also:

- About Panels (p. 78)

2.2.13. Cache Panel

The Cache panel is for more advanced users of Rocky. It is designed to list the Rocky processes that take up memory (Figure 1), and also enables you to remove any unused processes that are no longer needed (Figure 2). This information can be used to optimize how Rocky uses the memory available.

Note:

Memory allotment for Rocky is set from the Preferences dialog. (See also About Setting Global Preferences (p. 227).)

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

Figure 2.17: Cache Panel showing currently unused items in red

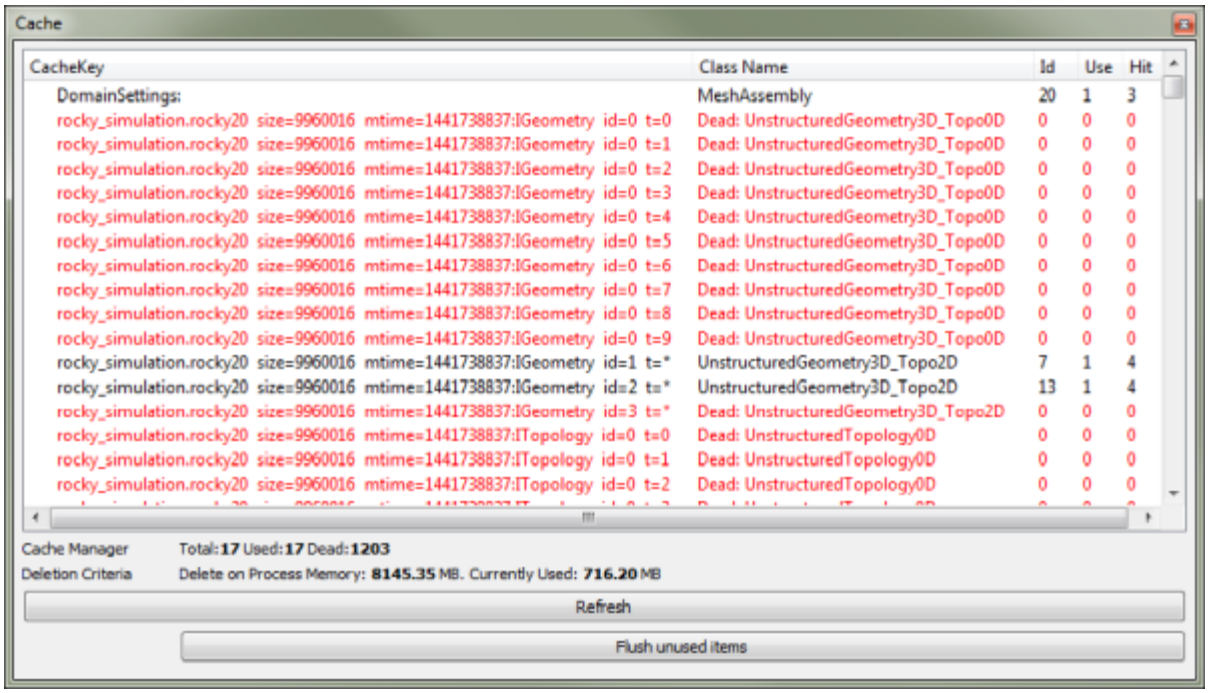
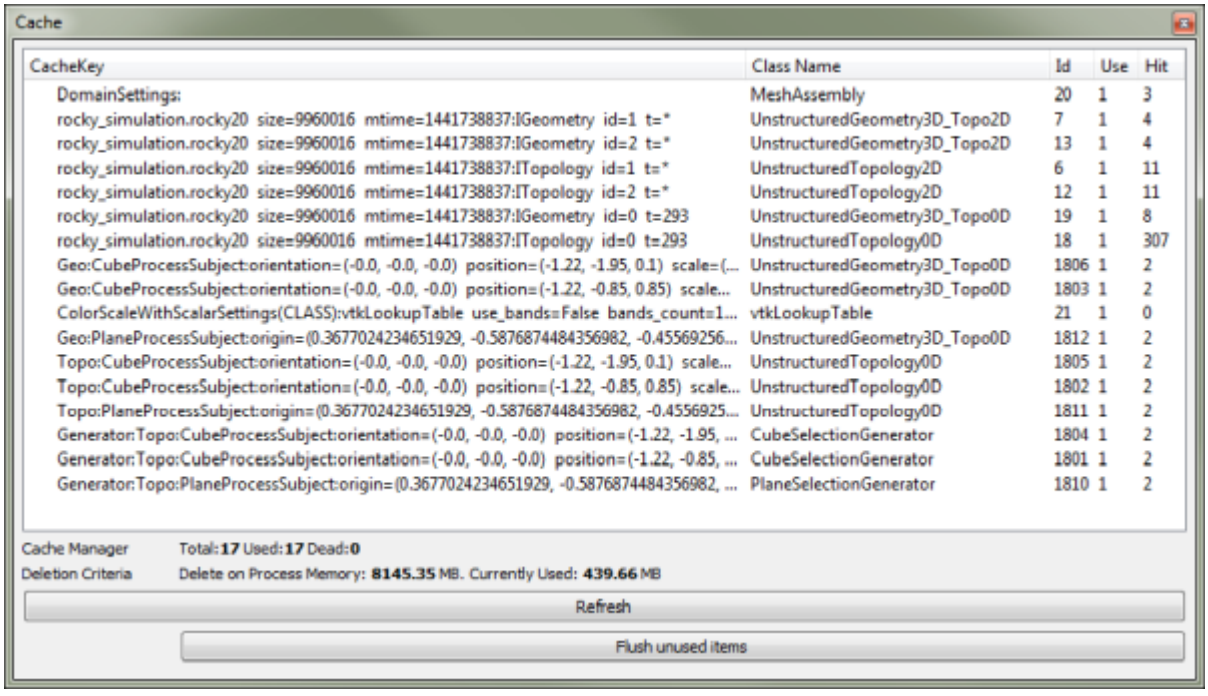


Figure 2.18: Cache Panel after unused items have been removed



Use the table below and images above to help you understand more about the Cache panel.

Table 1: Cache panel settings

Setting	Description
Cache list	Lists individual Rocky processes taking up memory. Tip: Click the **Refresh** button to view the current list.

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

Setting	Description
Cache Manager	Counts the processes in the Cache list and organizes them by the following criteria: • **Total**: The total number of cached items in the list. • **Used**: The number of cached items using memory that are currently in use. For example, triangles on a conveyor being visualized for the current output time. • **Dead**: The number of cached items using memory that are *not* currently in use. For example, triangles on a conveyor for a previously visualized output time. Though not currently in use, if you return to that output time, those triangles are resurrected from memory.
Deletion Criteria	Describes how processes will be deleted. Specifically: • **Delete on Process Memory** shows the maximum amount of memory Rocky is allocated. After that amount is used Rocky will start disabling processes to stay under the maximum. Note: Memory allotment is set on the Preferences dialog. (See also About Setting Global Preferences.) • **Currently Used** shows the amount of memory Rocky processes are currently using. Tip: Click the **Flush unused items** button to remove dead items from the list.

See Also:

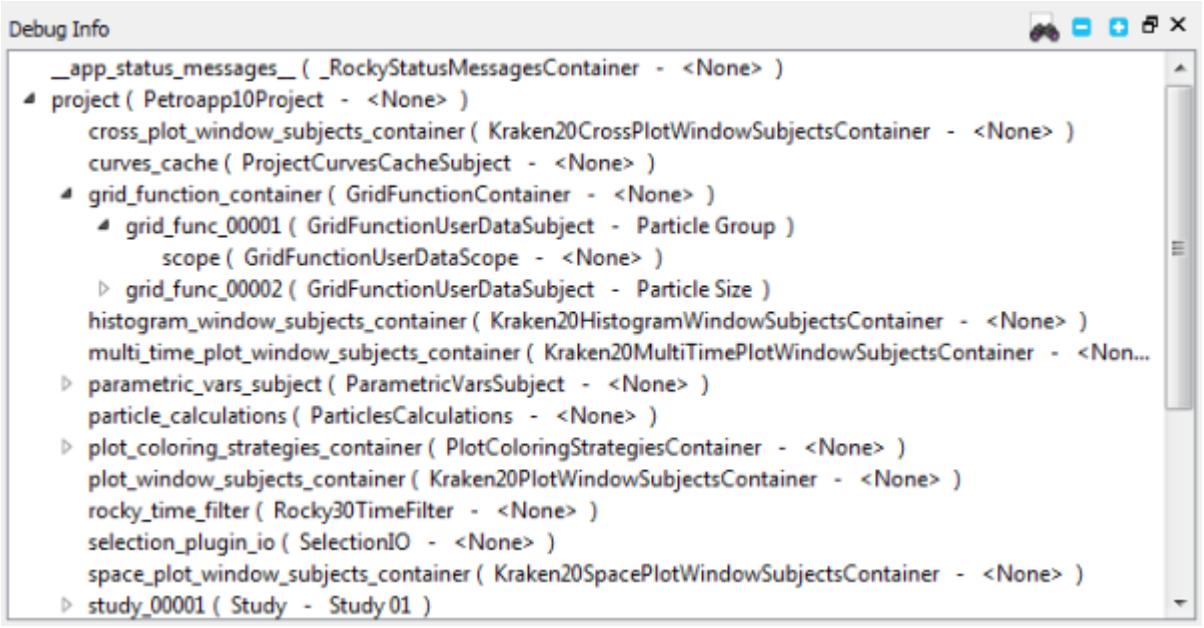
- About Panels (p. 78)

2.2.14. Debug Info Panel

The Debug Info panel is for advanced users of Rocky. It is designed to list code-related items for the purpose of solving issues found within the program (Figure 1).

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

Figure 2.19: Debug Info panel



See Also:

- About Panels (p. 78)

2.2.15. Undo/Redo History Panel

The **Undo/Redo history** panel Figure 2.20: Undo/Redo history panel showing some example actions (p. 107) is one way you can review and undo the actions you took on your project. Each separate step you take or setting you change is chronologically listed in this panel. When you Undo or Redo an action- either by using the panel, the toolbar, or keyboard - the choices you make are reflected here. The last action applied to the project is marked with a blue arrow; active actions are listed in black; inactive (or "undone") actions are listed in gray Figure 2.21: Undo/Redo history panel showing some example actions that have been undone (p. 107).

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

Figure 2.20: Undo/Redo history panel showing some example actions

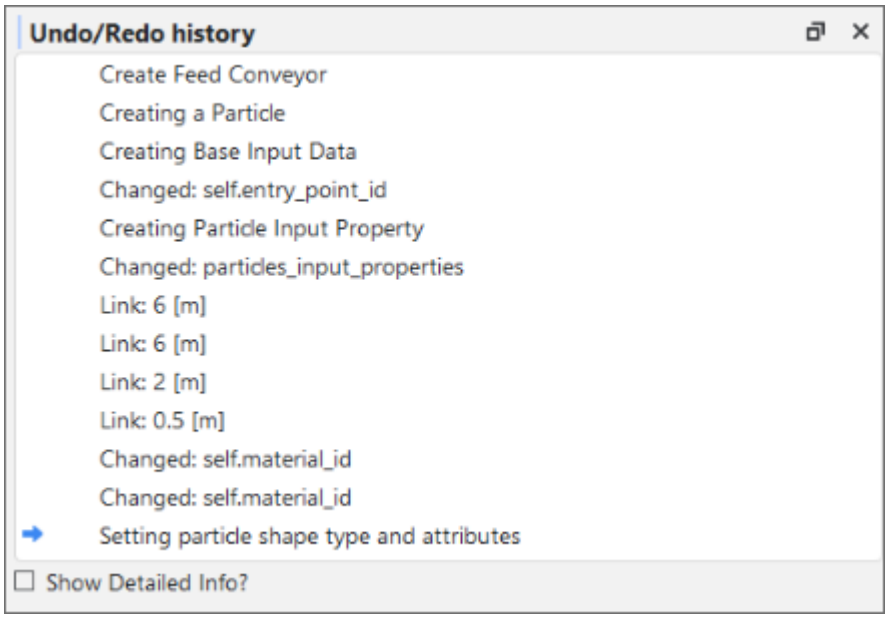
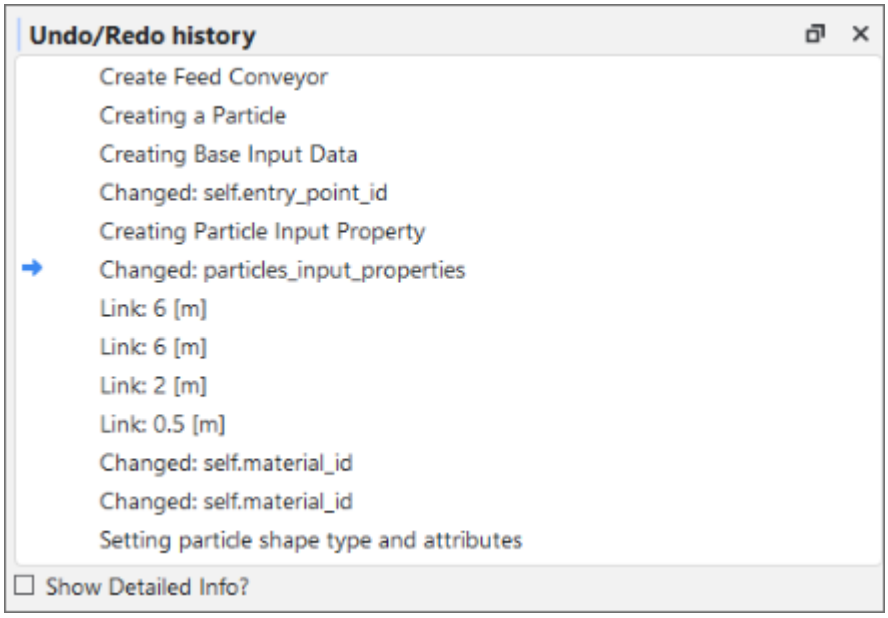


Figure 2.21: Undo/Redo history panel showing some example actions that have been undone



You can use the toolbar or keyboard to undo or redo one action at a time. You can use the panel to undo or redo every action up to the action you select. In this way, you can "undo" or "redo" as many actions as you like with only a double-click.

The actions listed in the panel are cleared every time you close and open the project.

What do you want to do?

- Undo or Redo Several Actions at Once (p. 108)
- Undo or Redo a Single Action at Once (p. 108)

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

UNDO OR REDO SEVERAL ACTIONS AT ONCE

1. Ensure the **Undo/Redo history** panel is open. (From the **View** menu, click **Undo/Redo history**.)
2. Do one of the following:
   - To undo several actions at once, find the blue arrow marking the current action, scroll up the list and then double-click the black-colored action to which you want to revert. The blue arrow now marks the action you selected and all later actions are colored gray to indicate that they are now inactive.
   - To redo several actions at once, find the blue arrow marking the current action, scroll down the list and then double-click the gray-colored action to which you want to redo. The blue arrow now marks the action you selected and all previous actions are colored black to indicate that they are now active.

See Also:

- About the Undo/Redo History Panel (p. 106)
- Undo or Redo a Single Action at Once (p. 108)
- View Frequently Used Tasks (p. 371)

UNDO OR REDO A SINGLE ACTION AT ONCE

- Do one of the following:
  - Click the **Undo** or **Redo** buttons on the toolbar.
  - Press **Ctrl+Z** or **Ctrl+Y** on your keyboard.

See Also:

- Undo or Redo Several Actions at Once (p. 108)
- About the Undo/Redo History Panel (p. 106)

2.2.16. About Workspaces

A Workspace is a collection of Rocky windows (p. 1396). Located in the center of the Rocky screen, the windows that make up a Workspace are where you choose to view the physical components of your simulation setup (3D View windows), and interact with the graphical results of your simulation (plot (p. 1393) and histogram windows).

You can have multiple Workspaces for different purposes, such as by analysis type. These are designated by Workspace tabs along the bottom of the Workspace area Figure 2.22: Example tabs at the bottom of the Workspace area (p. 109), which can also be renamed to help you better categorize your data Figure 2.23: Renaming a Workspace (p. 110). You can also use the tiling options in the Windows menu to better organize your windows on the Workspace Figure 2.23: Renaming a Workspace (p. 110).

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

Figure 2.22: Example tabs at the bottom of the Workspace area

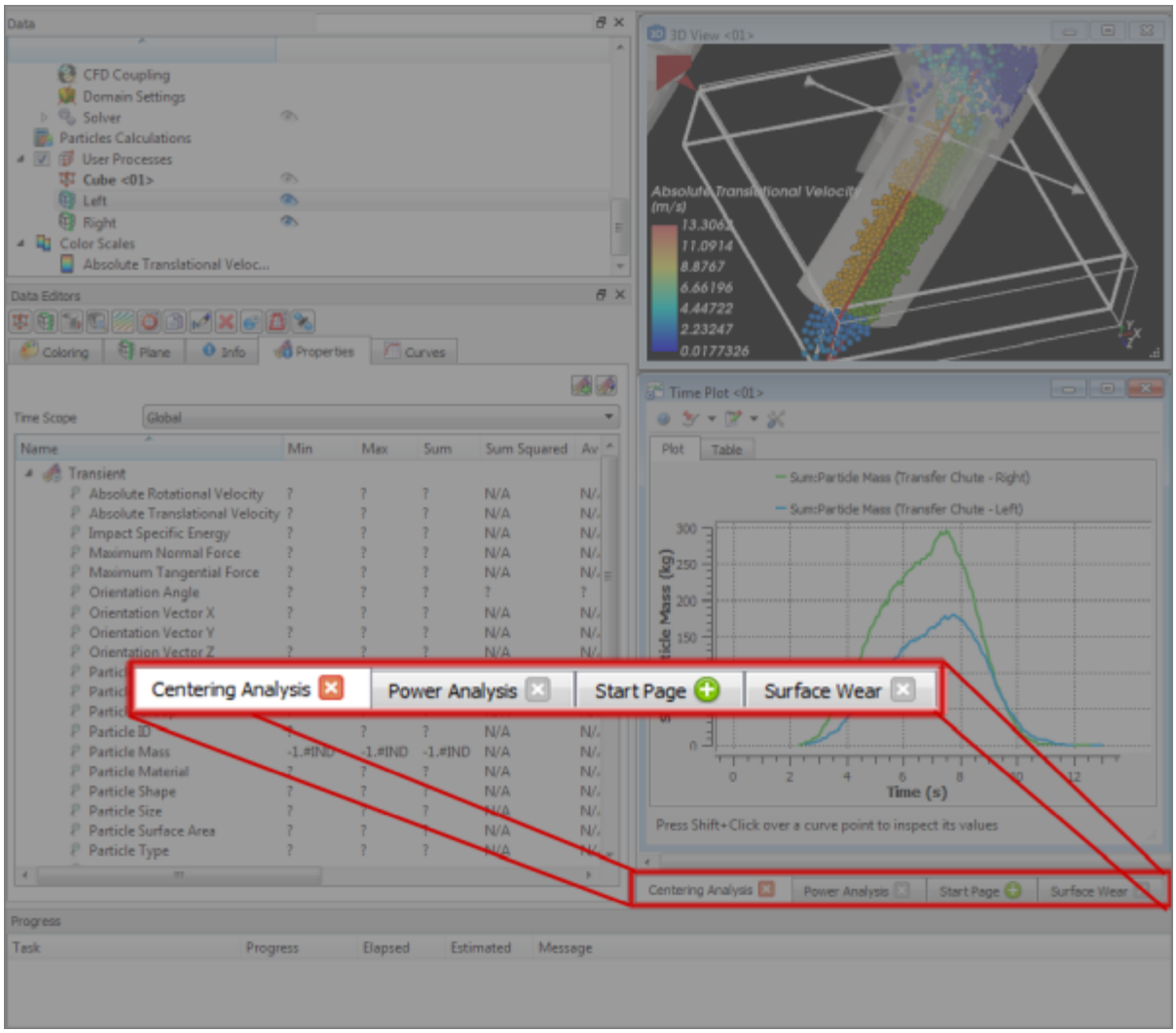


Figure 2.23: Renaming a Workspace

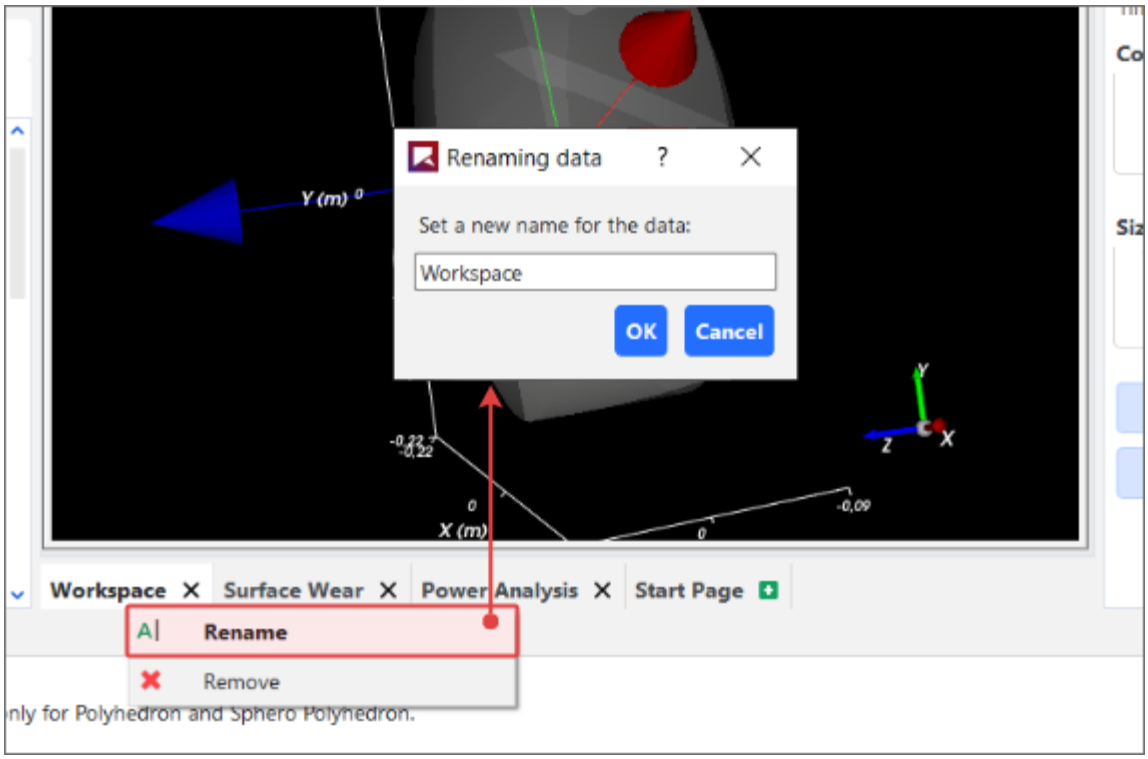
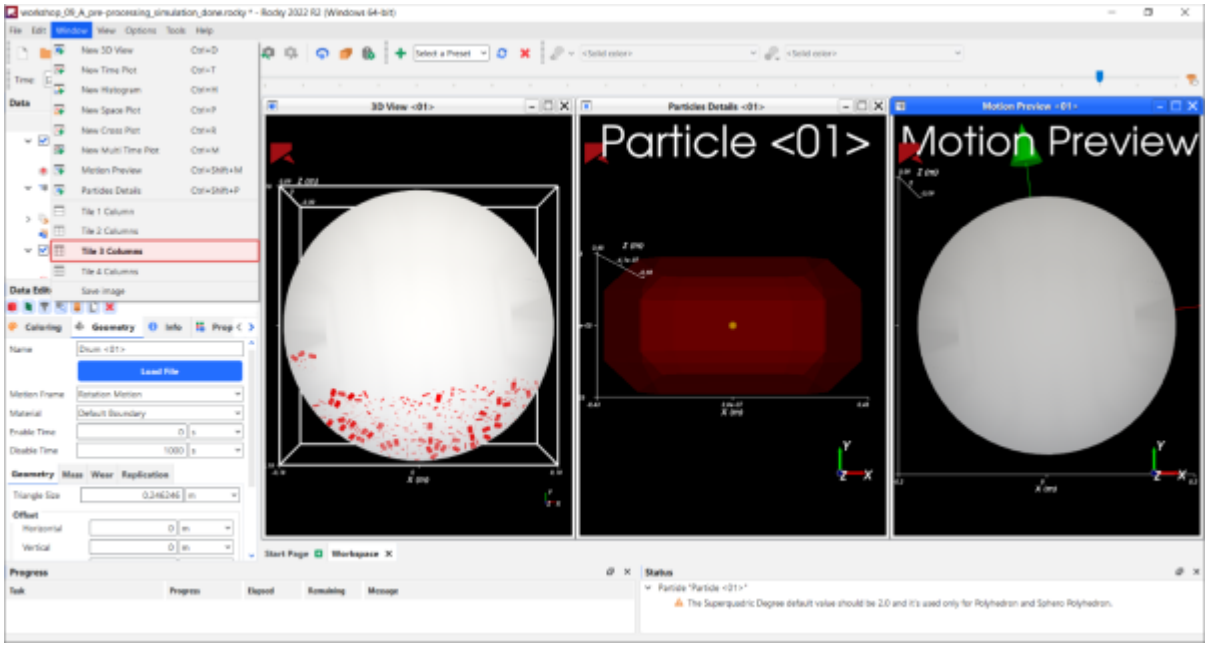


Figure 2.24: Tiling the windows on a Workspace by 3 columns



What would you like to do?

- Create a New Workspace (p. 111)
- Edit or Remove a Workspace (p. 111)

See Also:

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

- Main UI Components (p. 74)
- Reuse 3D View Window, Workspace, and User Process Setups in Other Projects (p. 403)

CREATE A NEW WORKSPACE

- From the bottom of the center Workspace area, on the **Start Page** tab, click the green "+" icon. A new tab appears at the bottom of the Workspace.

Tips:

- To change the name of the Workspace, right-click the Workspace tab and then in the **Renaming data** dialog, enter a new name and then click **OK**.
- To add windows to the Workspace, see the following topics:
  - Create and Modify a 3D View (p. 1040)
  - Graphing (Plot or Histogram) a Data Set Within Rocky (p. 1101)
  - Preview a Particle Shape in 3D (p. 681)
  - Preview a Motion in 3D (p. 573)

See Also:

- Edit or Remove a Workspace (p. 111)
- About Workspaces (p. 108)

EDIT OR REMOVE A WORKSPACE

Do one or more of the following:

- To remove an existing Workspace, right-click tab for the Workspace you want to remove, and click the red "X" icon. **Note**: Removing the Workspace also removes any window(s) (p. 1396) that you had placed on the Workspace. This can be reversed with the Undo function. (See also Undo or Redo a Single Action at Once (p. 108).)
- To add another Workspace, on the **Start Page** Workspace tab, click the green "+" icon.
- To change the name of the Workspace, do the following:
  1. Right-click the Workspace tab, and click **Rename**.
  2. From the **Renaming data** dialog, enter a new name for the Workspace, and then click **OK**.
- To arrange the windows orderly on a Workspace, from the Windows menu, select the **Tile Columns** option you want.
- To add windows to the Workspace, see the following topics:
  - Create and Modify a 3D View (p. 1040)
  - Graphing (Plot or Histogram) a Data Set Within Rocky (p. 1101)

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

See Also:

- About Workspaces (p. 108)
- Create a New Workspace (p. 111)

2.2.17. Other UI Functionalities

This section covers the not-so-obvious but useful features scattered throughout the Rocky UI and how it might benefit you to use them.

What would you like to do?

- Name Components to Affect Sort Order (p. 112)
- Show/Hide Components by Using Eye icons and Checkboxes (p. 113)
- See Input/Output Relationships in the Data Panel When Component Names are Bold (p. 114)
- Determine the Frequency of Calculations by Pinning/Unpinning Items (p. 115)
- Recognize Data Placeholders (p. 116)
- Double-Click the Status Panel to Jump to the Appropriate UI Location (p. 117)
- Use Colored Text to Validate the Syntax (p. 117)
- Drag/Drop Items into Windows (p. 117)
- Access the Right-Click Menu for Additional Features (p. 118)
- Use the Data Panel Search Bar to Quickly Find Components (p. 118)
- Refer to the Rocky Title Bar for Simulation Progress Details (p. 119)
- Lock a Curve Selection to a Plot to Quickly Change the Data Shown (p. 120)
- Double-Click a Plot Line, Bar, or Point to Change Its Appearance (p. 121)
- Recognize Shared Parameters with Asterisks (*) (p. 124)
- Use Colored Directional Handles (p. 126)

See Also:

- Understand the User Interface (p. 74)

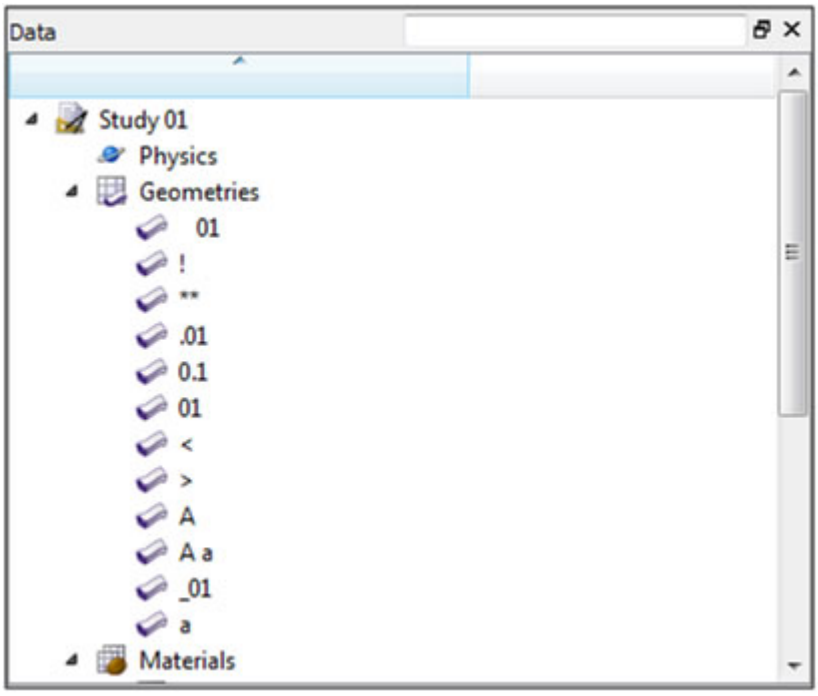
NAME COMPONENTS TO AFFECT SORT ORDER

Sorting in Rocky follows the rules for Extended ASCII character codes (For a full list, see Appendix B: ASCII Printable Characters (p. 1304).) In general, Names beginning with spaces are sorted first, followed by most special characters, followed by numbers, followed by uppercase letters, and ending with lowercase letters.

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

For example, Rocky would sort the following 12 geometry components in the order shown in Figure 1.

Figure 2.25: Example sort order in Rocky



See Also:

- Hidden Features (p. 112)

SHOW/HIDE COMPONENTS BY USING EYE INCONS AND CHECKBOXES

Rocky provides two complementary methods for determining whether a component or family of components will be shown or hidden in the Workspace (p. 1396): The eye icon (Figure 1), which affects the visibility of only that individual component, and the checkbox (Figure 2), which affects the visibility of all sub-components beneath the object.

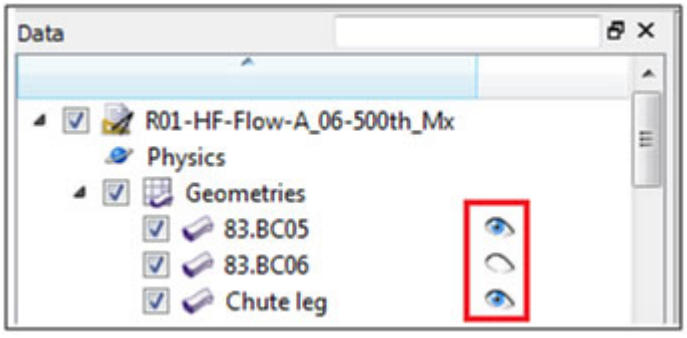


Figure 1 : Open and closed eye icons to the right of objects in the Data panel

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

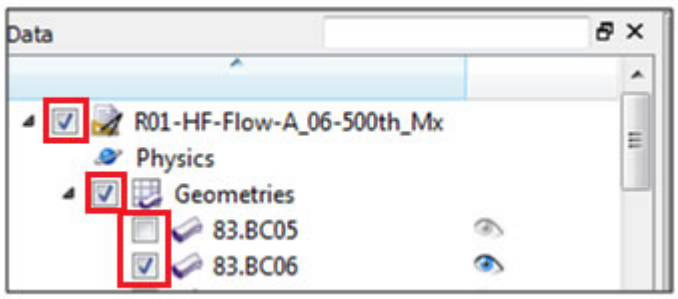


Figure 2 : Checkboxes to the left of objects in the Data panel

Eye Icon

An "open" eye icon (default setting) indicates that particular object is visible or shown in 3D View windows (p. 1396), plots (p. 1393), or histograms. A "closed" eye icon keeps that item in the simulation's calculations but makes the object appear invisible or hidden in the window. This is useful for seeing behind one object to another in an animation, for example, or for focusing on a particular curve (p. 1390) in a plot.

Checkbox

In the Data panel (p. 1393), the checkbox has the same visibility affect as the eye icon but is hierarchical, meaning it affects all children components directly beneath that item in the Data panel. For example, clearing the checkbox to the left of Geometries hides all individual geometry components listed beneath it.

In the Windows panel, the checkbox acts like the eye icon does in the Data panel: it shows/hides the individual object—in this case, various windows in the Workspace. Unlike in the Data panel, checkboxes in the Windows panel are not hierarchical.

See Also:

- Hidden Features (p. 112)

SEE INPUT/OUTPUT RELATIONSHIPS IN THE DATA PANEL WHEN COMPONENT NAMES ARE BOLD

Linked objects refer to the ability of a geometry component, Particles, or 1-Way LBM to provide data to a particular User Process, thus establishing an input/output relationship between them. This linkage is illustrated in the Data panel by the name of the object providing the data (output) turning bold when the User Process that is accepting the data (input) is selected, as shown in Figure 1.

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

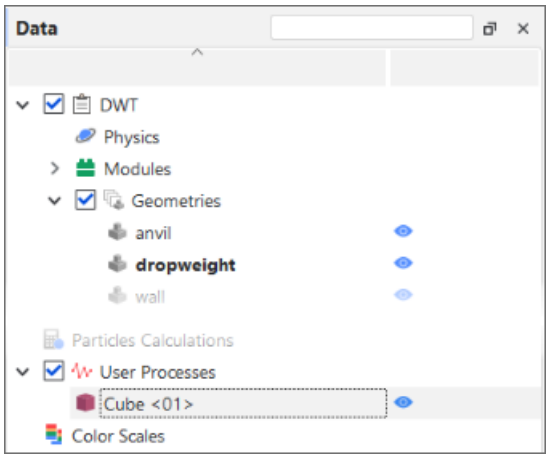


Figure 1 : Example of a geometry component shown in bold when the user process linked to it is selected

To create this kind of input/output relationship, the User Process must be created directly from the object providing the data. In the example above, the **Cube <01>** user process was created by selecting the **dropweight** geometry and then creating a Cube user process from either the Data Editor or right-click menu.

See Also:

- Hidden Features (p. 112)

DETERMINE THE FREQUENCY OF CALCULATIONS BY PINNING/UNPINNING ITEMS

After processing your simulation, you can view various types of data on the **Properties** and **Curves** tabs for your simulation settings like particles and geometry components. Each of the items listed on these tabs has a pin icon to the left of it, as illustrated in Figure 1.

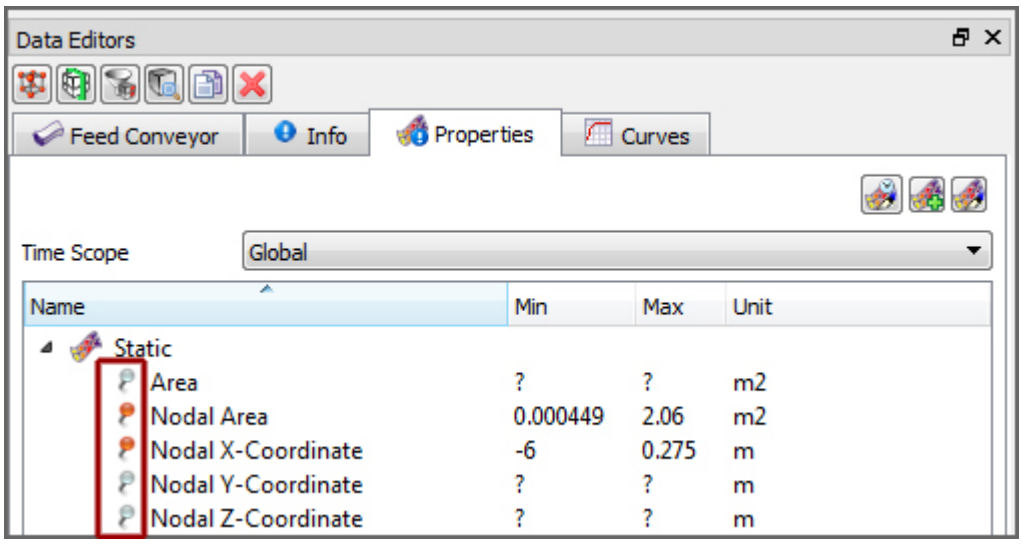


Figure 1 : Pin icons on the Properties tab of the Data Editors panel

The color of these pin icons indicates how often Rocky calculates the particular item, as follows

- A gray-colored pin (default) indicates that the item will not be calculated for any output time. Choosing to avoid calculating data that you don't need saves processing time and power during your simulation

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

- A red-colored pin indicates that the item will be calculated for each output time included in the simulation. And if settings affecting those values change, Rocky will recalculate to keep all the values current. **Important**: Choosing to have Rocky calculate the item at each output time is likely to take up a significant amount of processing time and power. Only choose to pin items for which having all output time calculations be kept up-to-date and available are truly necessary for your analysis.

Tip: To pin or unpin an item, just left-click the pin icon.

Alternately, you can choose to have Rocky calculate any values you are interested in once, by right-clicking the item and choosing **Compute Statistics**. (See also About Viewing an Individual Statistic (p. 1098).)

See Also:

- Hidden Features (p. 112)
- Use Properties and Curves (p. 931)

Recognize Data Placeholders

Rocky uses several different characters and strings as placeholders for calculations or explanations that would take up too much room in the UI. Here is a quick list of some placeholders you might run across and what they mean.

Note:

These placeholders might appear on the **Info**, **Properties**, or **Curves** tabs (**Data Editors** panel).

Placeholder	Description
?	Information isn't calculated yet. Same as "Not loaded."
Not loaded	Information isn't calculated yet. Same as "?".
-	Item contains no unit for the value displayed. Generally indicates a ratio, such as a percentage.
Unable to calculate	Not enough information available at this particular output time in order to calculate the values.
Unknown	Not enough information is known to calculate the value.
<ind>	Item contains no units for the value displayed. Generally indicates a index, such as a count or a coordinate position.
-1.#IND	Calculation is invalid. This can happen to Particle properties, for example, if there are no particles in the simulation at the time specified for the calculation.
N/A	Value is Not Applicable (N/A) to the selected item.

See Also:

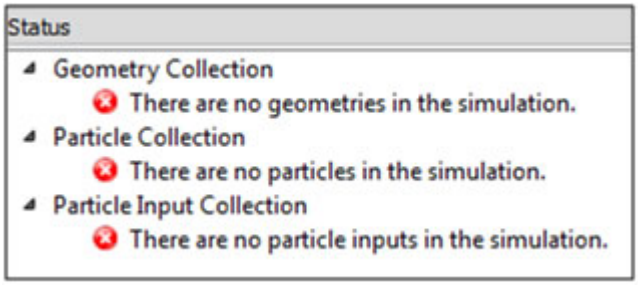
- Hidden Features (p. 112)
- About Properties and Curves (p. 933)

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

DOUBLE-CLICK THE STATUS PANEL TO JUMP TO THE APPROPRIATE UI LOCATION

If you have errors listed in the Status panel (Figure 1), you can double-click those errors to be taken to the location in the Data panel where you would address them.

Figure 2.26: Example of errors shown on the Status panel



For example, if you double-click the "There are no geometries in the simulation" error, Rocky would enable the **Geometries** item in the **Data** panel.

See Also:

- Hidden Features (p. 112)

USE COLORED TEXT TO VALIDATE THE SYNTAX OF YOUR ENTRIES

Rocky provides instant feedback for entering functions and variables into a parameter text field by coloring the text as you type, as explained below:

- Red text indicates that the syntax of the active entry is not yet valid. The syntax is based upon the rules of the Python programming language. It will tell you whether you have an open parenthesis that needs to be closed, for example.
- Green text indicates that the syntax of the active entry is valid. **Note**: Green text does not indicate that the value, function, or variable of active entry has been formatted correctly or is itself a valid entry. Validation happens after either pressing Enter or clicking away from the text field.
- Black text indicates a currently inactive entry.

See Also:

- Hidden Feature (p. 112)
- Customize and Extend Rocky with Python (p. 294)
- I briefly see long numbers when pasting in copied values (p. 1323)

DRAG/DROP ITEMS INTO WINDOWS

There are several tasks in Rocky that can be accomplished more easily by dragging items with your mouse and dropping them to another location, as described below:

- **Creating new views of Particles, individual geometry components, or Air Flow:** From the **Data** panel, select the component(s) you want to view and then drag and drop them to a window of your choice (3D View, Histogram, or Time Plot). Only the component(s) you selected will be added.

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

- **Creating new views of data sets**: From the **Data Editors** panel, on the **Curve** or **Properties** tab, select the data item(s) you want to show and then drag and drop them into the window type you want (3D View, plots or histograms).
- **Creating multiple Multi-Time plots in a single window**: After adding your first data set, hold the Ctrl key while selecting your next data set and then drag and drop it into the gray area of an existing Multi-Time plot window. Another plot appears in the same window.

Tip: You can also multi-select items to drag and drop. (For example, show multiple geometry components at once in a 3D View.) Just hold the Ctrl or Shift key while left clicking your selections with the mouse, and then drag and drop as usual.

See Also:

- Hidden Features (p. 112)

ACCESS THE RIGHT-CLICK MENU FOR ADDITIONAL FEATURES

There are a few things in Rocky that can only be accomplished by right-clicking the item, as described below:

- **Adding/Importing geometry components**: From the **Data** panel, right-click **Geometries** to access the menu that enables you to add or import new geometry components.
- **Computing statistics for only the current output time**: From the **Data Editors** panel, on the **Curves** or **Properties** tab, right-click the data item you want to view values for and then click **Compute Statistics** from the right-click menu.
- **Showing only a few components in a new window**: From the **Data** panel, select the item(s) you want displayed in a new window, right-click your selection, and then from the right-click menu, point to **Show in New** and then click the window type you want. Only the items you selected appear in the new window.
- **Import your own User Process shape for particle analysis**: From the **Data** panel, right-click **Particles**, point to **User Processes**, and then click **Polyhedron (Envelope)**. From the **Select the STL file for the polyhedron** dialog, navigate to the STL file you want, and then click **Open**. The polyhedron you chose appears under the **User Processes** list. **Note**: You can also create a custom-shaped User Process from any existing User Process that was originally created from a Particle set. (See also See Input/Output Relationships in the Data Panel When Component Names are Bold. (p. 114)) From the **Data** panel, under **User Process**, right-click a User Process that was created from a Particle set and then follow the instructions above.

See Also:

- Hidden Features (p. 112)
- Add and Edit Geometry Commponents (p. 480)
- About Viewing an Individual Statistic (p. 1098)
- Filter Views and Data with User Processes (p. 1174)

USE THE DATA PANEL SEARCH BAR TO QUICKLY FIND COMPONENTS

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

The search bar is located at the very top right of the Data panel, as shown in Figure 1.

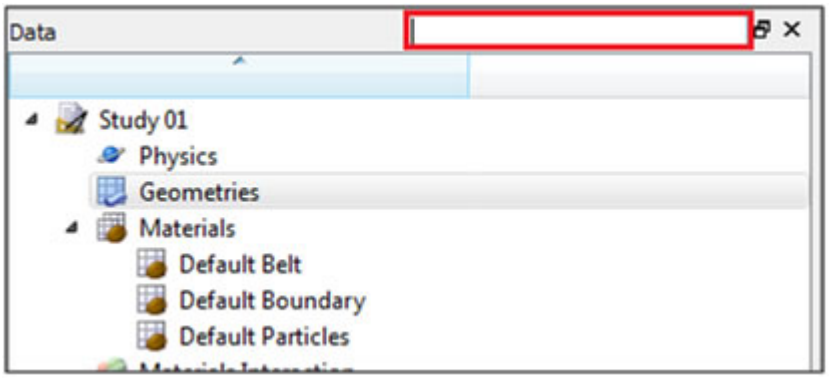


Figure 1 : Data panel search bar

When you start typing into it, Rocky immediately displays in the Data panel the entire hierarchy of the component(s) that best match what you typed. In a simulation setup with many different components, using the search bar can help you find what you need faster than scrolling.

Tips:

- To search the names in order from left to right, enter the first few characters of the component you want to find. *For example, to find all components that begin with "belt," type belt into the search box. You would find "belt 01" and "belt 02" but not "feeder belt" by using this method.*
- To search anywhere in the name, type an asterisk (*) followed by the characters anywhere within the name that you want to find. *For example, to find all components containing "belt" anywhere in the name, type *belt into the search box. In this way, you would find the "feeder belt" component as well as "belt 01" and "belt 02."*

See Also:

- Hidden Features (p. 112)

REFER TO THE ROCKY TITLE BAR FOR SIMULATION PROGRESS DETAILS

When you are actively processing a simulation (see also About Starting a Simulation (p. 908)), the Title bar for the Rocky program displays useful details about the simulation's progress. (Figure 1.)

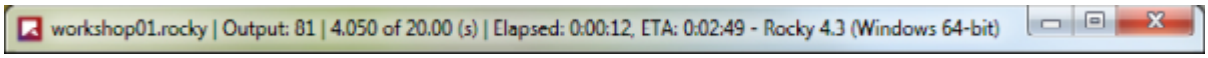


Figure 1 : Example of Title bar progress information shown while a simulation is processing

This progress information in the Title bar includes the following:

- **Output**: The total number of output files that have been saved thus far. In the example in Figure 1, the total number of output files (p. 1393) saved so far is 81.
- **Current progress**: The particular second of the total Simulation Duration that Rocky is currently calculating. Note that Rocky calculates in finer detail than is typically determined by the Output Frequency in which it saves output files. In the example in Figure 1, although Rocky is currently calculating for second 4.050, because the simulation output frequency is set to 0.05 seconds, it won't actually save a 82nd output file representing 4.100 seconds until the current progress reaches that point.

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

- **Elapsed**: Amount of real time since starting (or resuming) the simulation processing. In the example in Figure 1, 12 seconds have elapsed since the simulation was resumed.
- **ETA**: Rocky's estimate of how much real time the simulation will take to complete. This is a rough approximation Rocky extrapolates based upon the amount of time the last several outputs took to calculate. In the example in Figure 1, Rocky estimates that it will take roughly 2 minutes and 49 seconds to complete. The ETA will get longer as more and more particles enter the simulation and it takes more time for Rocky to calculate each output. Should the simulation reach a point where more particles leave the simulation than enter it, the ETA will begin to get shorter. Once the simulation enters a steady state, this value will likely be a very good estimate of the remaining time.

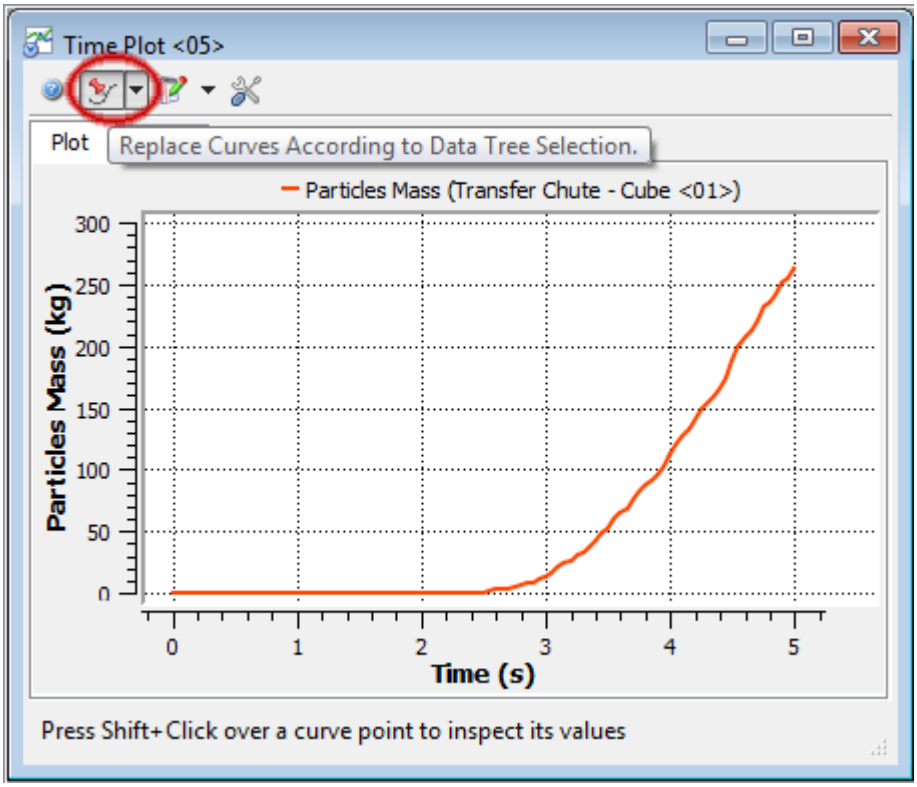
See Also:

- Hidden Features (p. 112)
- About Solver Parameters (p. 464)

LOCK A CURVE SELECTION TO A PLOT TO QUICKLY CHANGE THE DATA SHOWN

If you want to see the same curve (for example, Particle Mass) in a plot (for example, Time Plot) across many similar Data panel elements (for example, several User Processes created from Particles), you can use the **Replace Curves According to Data Tree Selection** toggle button (Figure 1), which is available for any Time Plot, Multi Time Plot, or Cross Plot.

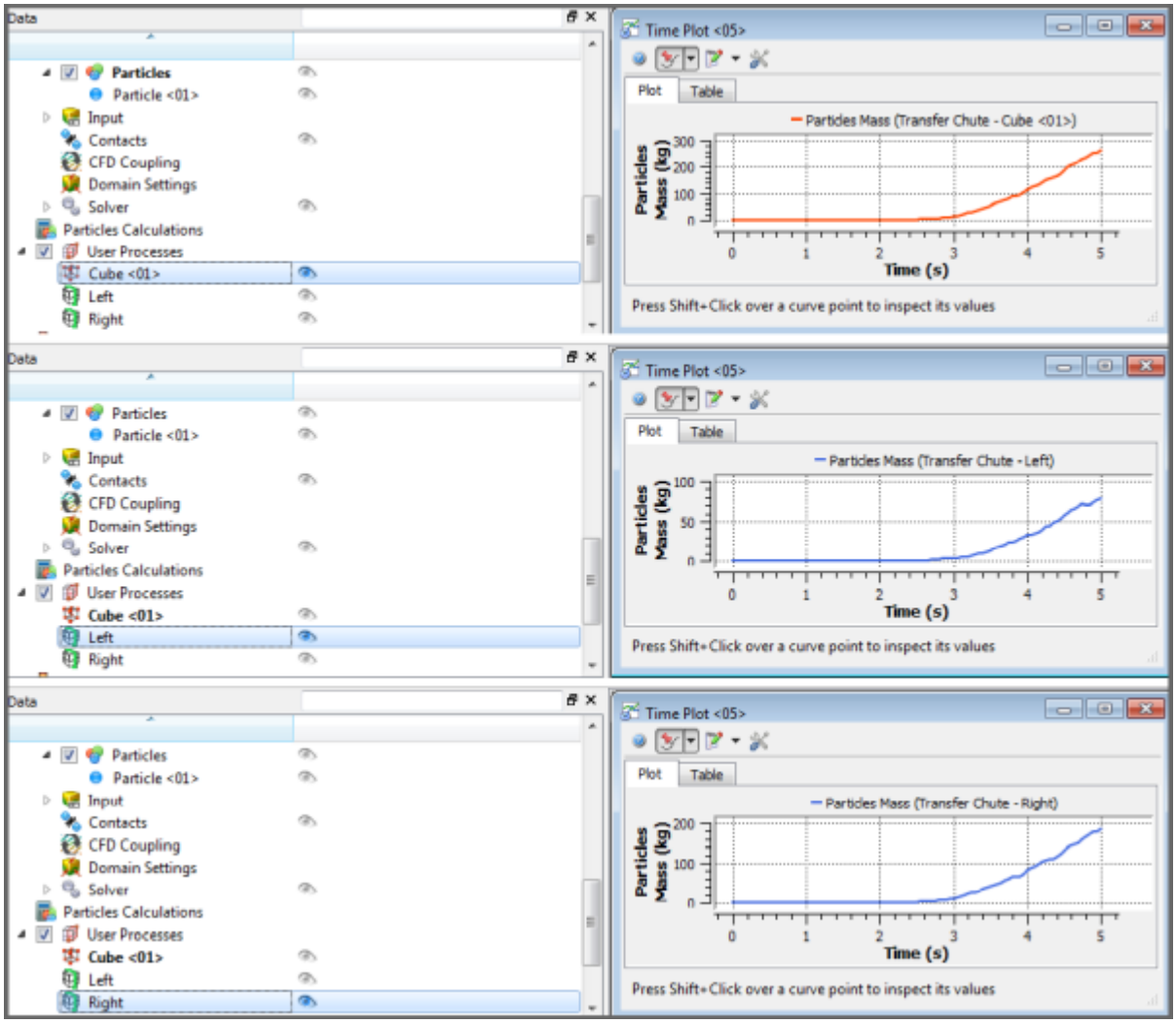
Figure 2.27: Location of toggle button



When you toggle (push in) this button on a plot you have created and then select another similar item on the Data panel, the curve you original selected stays the same but the data is replaced with the data corresponding to the item you just selected (Figure 2).

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

Figure 2.28: Original plot (top); second Data panel option selected with button toggled (middle); third Data panel option selected with button toggled (bottom)



In this way, you can quickly compare the same curve across many similar Data panel items without having to create many separate plots. This button also works if you multi-select curves.

To turn off this feature, click the toggle button again to push it out.

See Also:

- About Curves (p. 989)
- About Graphing (Plot or Histogram) a Data Set Within Rocky (p. 1101)

DOUBLE-CLICK A PLOT LINE, BAR, OR POINT TO CHANGE ITS APPEARANCE

Similar to the **Edit Expression** dialog that you get when you click the **Edit Formula** button on the **Table** tab of a Time Plot (see also About Time Plots (p. 1102)), you can also change the Pen style, color, and width as well as the data point appearance by double-clicking the line, bar, or point in the plot

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

or histogram. This will bring up the **Edit Curves** dialog (Figure 1) and from there, you can change the settings as you like.

Tip:

You can also right-click the line, bar, or point and then from the menu, select the **Edit** option pertaining to the curve you want to change. This will bring up the same **Edit Curves** dialog you get when you double-left-click (Figure 1).

See the image, table, and procedure below for information about the Edit Curves dialog.

Figure 2.29: Edit Curves dialog for a Time Plot

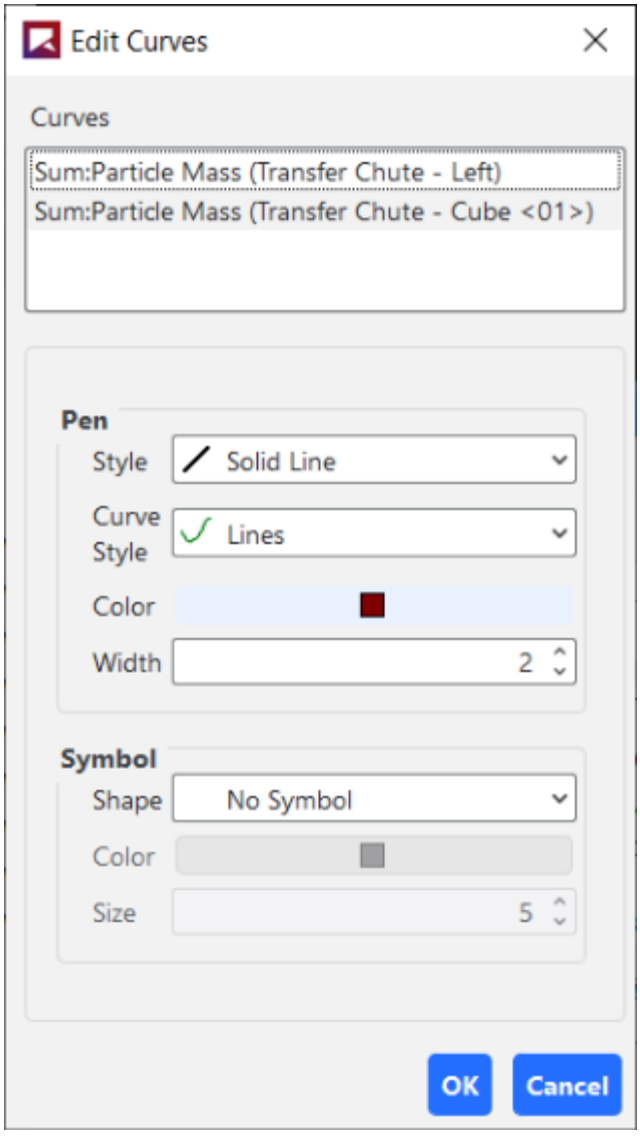


Table 1: Edit Curves dialog options

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

Setting	Description	Range
Curves	Enables you to select which data set(s) you want to change.	Options limited by the plot or histogram selected
Pen		
Style	For plots only (not histograms), this enables you to set what kind of line style you want represented for the selected data set. **Tip:** If you want the data set represented on the plot by symbols instead of lines, select **No Line** and then choose what you want under **Symbol**.	No Line; Solid Line; Dash Line; Dot Line; Dash Dot Line; Dash Dot Dot Line
Curve Style	For plots only (not histograms), when you select a **Style** other than **No Line**, this enables you to set how the data set is represented in the plot. What you set here will be drawn in the **Style** indicated.	Lines; Sticks; Steps
Color	For histograms, and also for plots with a **Style** other than **No Line**, this enables you to change the color of the bar or plot line for the selected data set.	Options limited by the **Select Color** dialog
Width	For plots only (not histograms), when you select a **Style** other than **No Line**, this enables you to change the width of the plot line for the selected data set.	Whole numbers greater than or equal to one
Symbol		
Shape	For plots only (not histograms), this enables you to add to the plot a symbol marking the curve's data point. For Time Plots and Multi Time Plots, a symbol will appear for each recorded output time. (For example, every 0.05s.) **Tip:** If you want the data set represented on the plot by lines instead of symbols, select **No Symbol** and then choose what you want under **Pen**.	Ellipse; Rectangle; Diamond; Triangle; Down Triangle; No Symbol
Color	For plots only (not histograms), when you select a **Shape** other than **No Symbol**, this enables you to change the color of the symbol used to mark the data points of the curve.	Options limited by the **Select Color** dialog
Size	For plots only (not histograms), when you select a **Shape** other than **No Symbol**, this enables you to change the	Whole, positive values

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

Setting	Description	Range
	size of the symbol used to mark the data points of the curve.	

TO CHANGE THE APPEARANCE OF A PLOT LINE, BAR, OR POINT:

1. Directly within the plot or histogram window, double-click the line, bar, or point that you want to change. The **Edit Curves** dialog appears.
2. From the **Curves** box, select the data set(s) you want to modify.
3. Under **Pen** and **Symbol**, select the options you want.
4. Click **OK**. The changes you made are reflected in the plot or histogram window.

See Also:

- About Changing the Appearance of a Graph (Plot or Histogram) (p. 1126)
- Graphing (Plot or Histogram) a Data Set Within Rocky (p. 1101)

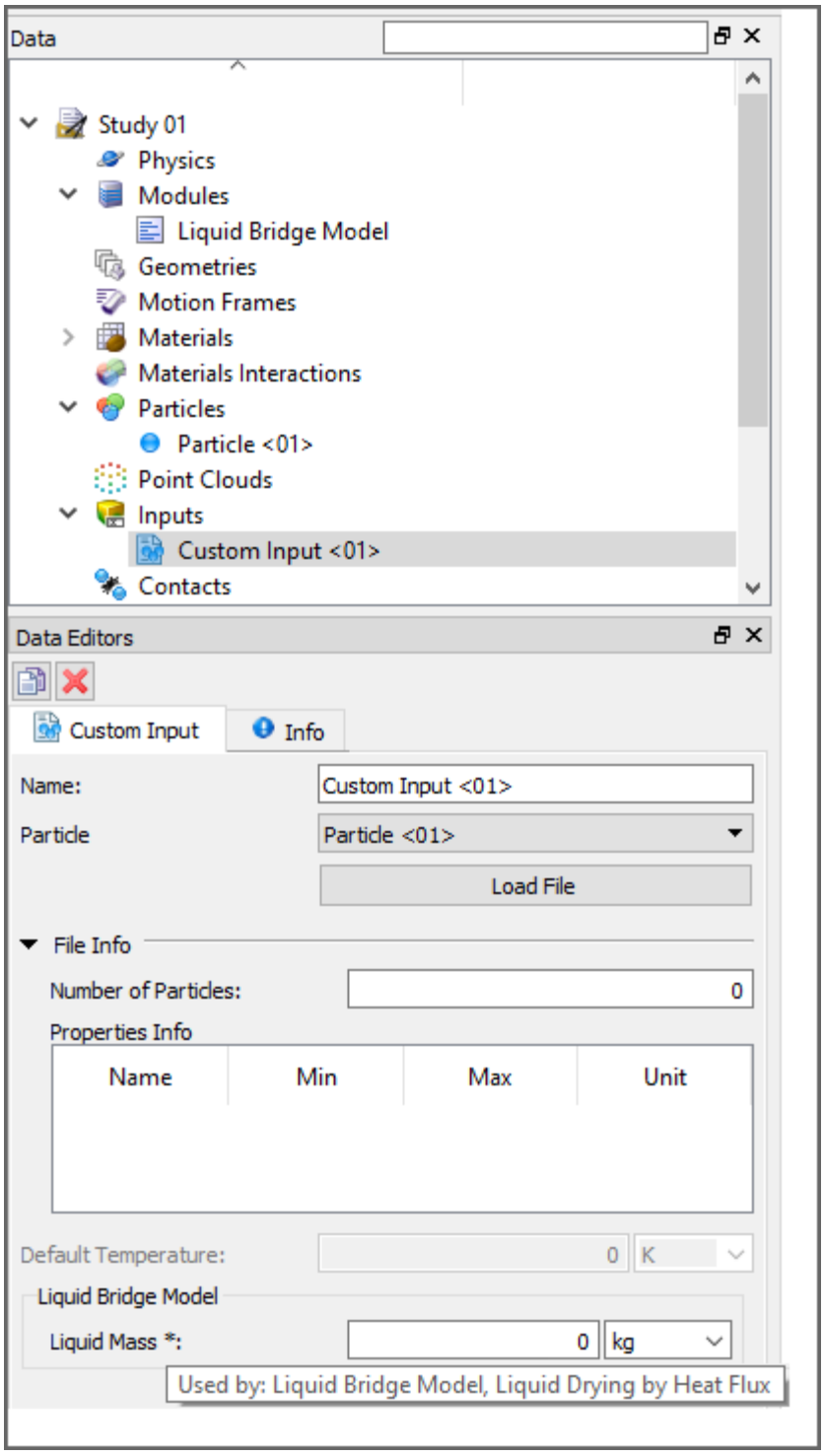
RECOGNIZE SHARED PARAMETERS WITH ASTERISKS (*)

Occasionally, Module (p. 1392) parameters in Rocky will be shared by parameters of other Modules. This means that a value entered in one location in the Rocky UI will be automatically duplicated in the other location.

To help you understand when this is happening, Rocky will place an asterisk (*) after the parameter name that is being shared. If you then hover your mouse cursor over the field for that asterisked parameter (Figure 1), the tooltip that appears will tell you what other Modules are sharing that same parameter.

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

Figure 2.30: Liquid Bridge Model Module settings showing a shared parameter



In the example shown in Figure 1, the asterisk following the name of the **Liquid Mass*** parameter indicates that there are one or more identical **Liquid Mass** parameter(s) being defined elsewhere in the Rocky UI. When you view the tooltip for this parameter's field, you learn that this parameter is shared by two separate Modules, this one included.

If you change the value for this shared parameter, it will automatically be updated in the other location(s).

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

Note: In this version of Rocky, the **Liquid Bridge Model** Module is provided as an external module (p. 1391). Refer to the Install an External Module (p. 223) topic for details.

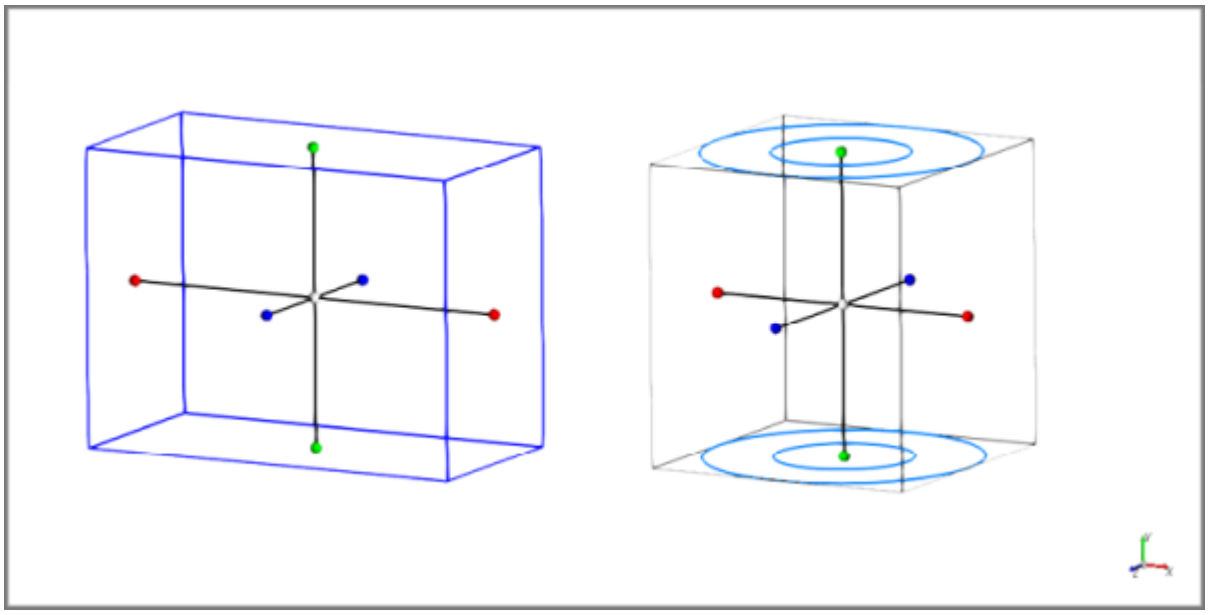
See Also:

- Hidden features (p. 112)
- About Modules Parameters (p. 431)

USE COLORED DIRECTIONAL HANDLES

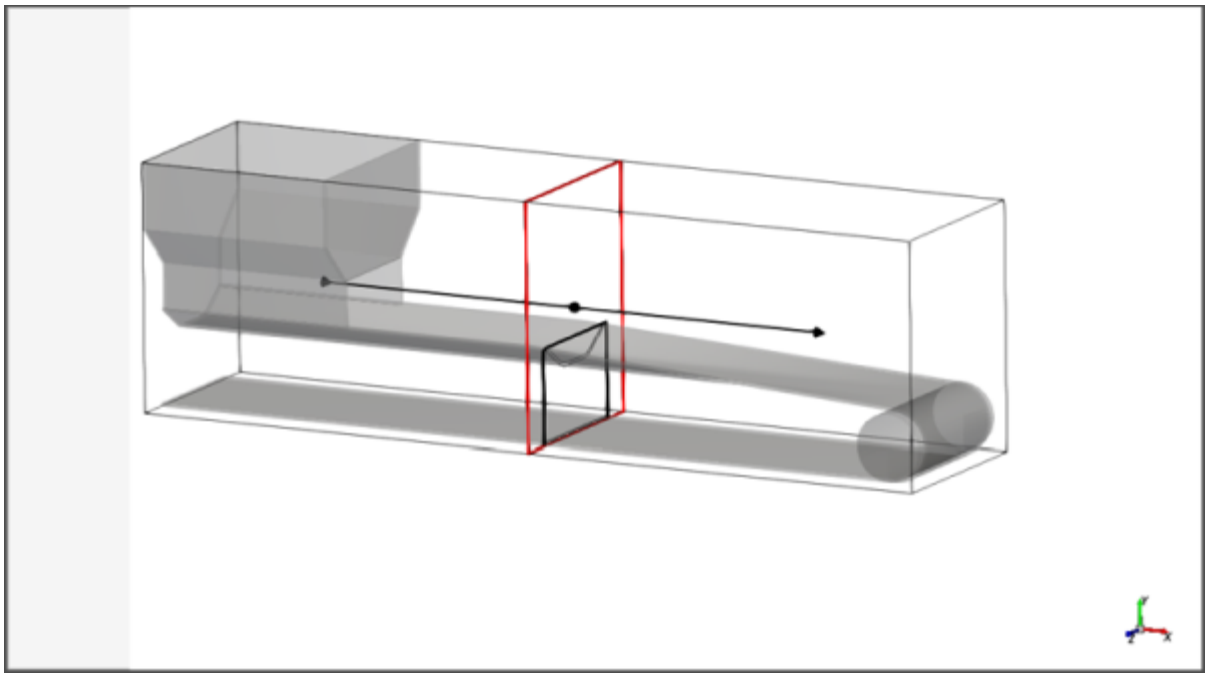
When viewed in a 3D View window, you can change the shape and location of some User Process, Volume Fill, and Region of Interest shapes by using the colored directional handles, as shown in Figures 1 and 2 below.

Figure 2.31: Directional handles for a Cube (left) and a Cylinder (right) Region of Interest



Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

Figure 2.32: Directional handles for a Plane User Process



HANDLE DEFINITIONS FOR SHAPES

For shapes like Cubes and Cylinders (Figure 1), the colors of the handles are defined in Table 1.

Table 1: Handle color definitions for shapes

Handle (Dot) Color	**Corresponds To**
Red	The local X axis.
Green	The local Y axis.
Blue	The local Z axis.
White	The center of the shape. **Note**: For Volume Fill shapes, the blue dot in the center is the Seed location but this can only be moved by defining the **Seed Coordinates**. (See also Create a New Volume Fill Input for Particles (p. 741).)

HANDLE DEFINITIONS FOR PLANES

For planes (Figure 2), the directional handles are colored according to the Font Color specified for the 3D View (see also About Using the Window Editors panel to Change the Selected 3D View (p. 1058)). The shapes of these colored handles are defined in Table 2.

Table 2: Handle shape definitions for planes

Handle Shape	**Corresponds To**
Dot	The location of the plane's origin.

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

Handle Shape	Corresponds To
Arrow	The direction of the plane's normal.

Using the Handles

Within the 3D View window, you can click and drag these handles to the new location you want.

Tip:

To move the whole shape without deforming it, do one of the following:

- Move only the center handle.
- Hold either the **Shift** or **Ctrl** key while you click and drag any of the outer handles with your mouse.

Or, for more precise control, you can enter exact values in the Data Editors panel.

See Also:

- Hidden Features (p. 112)
- About Regions of Interest (p. 522)
- About Eulerian Statistics (p. 1226)
- Analyze Data within a Defined Shape (p. 1175)
- About Adding and Editing Particle Inputs (p. 708)

KEYBOARD ACCESSIBILITY AND SHORTCUTS

While Rocky was not designed for full software accessibility, it does offer some limited features for keyboard control. For example:

- **TAB key**: Press to move from option to option within a Rocky dialog or panel.
- **SPACEBAR**: Press to select or clear a check box, or to select buttons in panels.
- **UP/DOWN ARROW keys**: Press to select list items within a Rocky dialog, Panel, or menu.
- **LEFT/RIGHT ARROW keys**: Press to view different menus or to move horizontally in Rocky dialogs.
- **ESC key**: Press to close a dialog.
- **ENTER**: Press to select a button within Rocky dialogs.

TO VIEW THE FULL KEYBOARD SHORTCUT LIST:

1. From the **Options** menu, click **Preferences**.
2. From the **Preferences** dialog, under **Properties**, click **Shortcuts**.

See Also:

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

- Understand the User Interface (p. 74)

FILE TYPES AND FOLDERS IN ROCKY FILE

There are several unique file types and folders used in Rocky, and several common ones. Use the tables below to identify when and why certain file types and/or folder locations are used.

Note: Only the simulation project file is saved to the directory location you choose. All other files created by Rocky (marked as "System" below) are saved to a project subfolder created automatically within the same directory as your project file. System files are critical for Rocky functioning and should not be edited, moved, or deleted.

Files you choose to export out of Rocky, including animation, image, and data files, are saved to the locations you specify. See the below tables for more detail.

Table 1: File Types Used in Rocky

Type	Description	Extension
Rocky Simulation Files		
Simulation project file (Rocky v4 or later)	Contains all parameters set and analyses completed for a particular simulation created in Rocky 4 or later, and is linked to the simulation output files (.rhs, .rhc, and .rhm) that are created during processing. • One project file is saved for each simulation. • Opening a project file will display the results from the output files (.rhs, .rhc, and .rhm) that were open when the project was saved.	.rocky
Simulation project file (Rocky v3 only)	Contains all parameters set and analyses completed for a particular simulation created in Rocky 3, and is linked to the simulation output files (.rhs) that are created during processing. **Tip**: Use the extension of the simulation project file to help you determine in which version of Rocky the project was created. Projects created in older versions of Rocky will have different features than those created in Rocky v4 or later versions.	.rocky30
(System) Simulation output file for particles and boundaries	The particle- and boundary-related results of the simulation that are created during processing. One output file is saved per simulation output frequency; many output files make up a whole simulation. • Output files are accessed by opening the project file (.rocky) file and then moving the **Time** toolbar slider to the location you want.	.rhs

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

Type	Description	Extension
	• Displaying an output file shows the image (in a 3D View or Particles Details window) or data (in a plot or histogram) for that particular moment in the simulation.	
(System) Simulation output file for contacts	The contact-related results of the simulation that are created during processing. This data is required to calculate particle and boundary movement and is by default saved for each output time to help make Resuming a Stopped Simulation faster. (See also Resume Processing a Stopped Simulation.) **Tip**: To reduce the size of your project files, clear the **Collect Contacts Data ** checkbox. (From the **Data** panel, click **Contacts** and then from the **Data Editors ** select the **Contacts** tab.) (See also About Contacts.)	.rhc
(System) Simulation output file for motions	The motion-related results of the simulation that are created during processing. One output file is saved per simulation output frequency; many output files make up a whole simulation. • Output files are accessed by opening the project file (.rocky) file and then moving the **Time** toolbar slider to the location you want. • Displaying an output file shows the image (in a 3D View or Motion Preview window) or data (in a plot or histogram) for that particular moment in the simulation.	.rhm
(System) Solver file	Used for storing information related only to the setup and calculated results of the simulation. Does not include post-processing information, such as views, graphs, or animations.	.rocky20
(System) Solver output files	System files containing runtime simulation information meant to be processed by the application. (See also About the Simulation Log File.)	.rocky20.log .rocky20.out .rocky20.prg
Rocky lock file	System file used to prevent another instance of the currently active project from being modified elsewhere. This file is created automatically by Rocky every time you open a project and remains on your system until you close the project, essentially locking the project file from being changed by anyone but you as long as you have the project open. This feature is especially useful in shared computing and server environments where multiple users may be contributing to the same Rocky projects but using different computing resources. It is for this reason that modifying, moving, or deleting the lock file is not recommended unless you are certain no other copies of the project are being worked on. (See also Rocky says my project file is "locked".)	.lock

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

Type	Description	Extension
(System) Current output file	This is where Rocky saves the initial and final output time value, which is used to determine the current output time.	_H.txt
Rocky script file	File used to contain *script* information, the latter of which includes recorded steps of a repeatable UI task. (See Creating and Using Your Scripts for more information.)	.py
Rocky archive file	File used to contain all simulation project and supporting folders necessary for sharing the simulation project and results with others, such as Rocky support. (See also Archive a Simulation Project.)	.rocky_archive
Rocky project context file	File used to contain the setup criteria for the 3D View windows, Workspaces, and User Processes used in the project. (See also Reuse 3D View Window, Workspace, and User Process Setups in Other Projects.	.rocky_template
Files Exported Out of Rocky		
Simulation statistics file	File created by exporting curves from a plot or histogram, thereby enabling the data to be viewed and/or edited in an external spreadsheet program, such as Microsoft Excel.	.csv
Animation file	File created by exporting an animation, thereby enabling the simulation to be viewed outside of Rocky. (See About Creating and Saving an Animation for more information.)	.avi
Image files	Used for saving or exporting a snapshot of what is currently being displayed in a window on the Workspace, or for importing into Rocky as a logo to be shown in the 3D View.	.png .jpg .bmp .pnp
Files Imported Into Rocky		
3D CAD model files	Used for importing geometries into Rocky. • XGL imports a multi-bodied geometry as individual components. The geometry component names are not retained. • STL imports a multi-bodied geometry as a single component. • DXF imports a multi-bodied geometry as individual components. The geometry component names are retained.	.xgl .stl .dxf .msh

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

Type	Description	Extension
	• MSH imports a multi-bodied geometry as individual components. The geometry component names are not retained. **Note:** Unlike in previous versions, none of these file types retain the component colors assigned in the CAD model upon import into Rocky.	
Fluent Case file	Used for importing geometries from an Ansys Fluent case file into Rocky. Geometry component names will be retained in this scenario. Also used to specify the Ansys Fluent simulation setup for 2-Way Fluent Coupling analyses. Contains the grid, boundary conditions, solution parameters and other data related to a fluid flow simulation problem.	.cas
Fluent Case file (compressed)	Compressed version of a Fluent Case file (CAS). Used both for importing geometries from an Ansys Fluent case file into Rocky, and also to specify the Ansys Fluent simulation setup for 2-Way Fluent Coupling analyses.	.cas.gz
Fluent Case file (HDF5)	Hierarchical Data Format (HDF) version of a Fluent Case file (CAS). In this version of Rocky, CAS.H5 files can be used both for importing geometries, and to specify the Ansys Fluent simulation setup for 2-Way Fluent Coupling analyses.	.cas.h5
Fluent Mesh file	Used for importing geometries from Ansys Fluent into Rocky. Geometry component names will not be retained when importing a MSH file into Rocky.	.msh
Ansys Motion Functional Mock-Up Unit file	Bundled file containing all motion setup information required for a 2-Way Coupled simulation with Ansys Motion, including geometries. As such, can be used for geometry import into Rocky. Currently supports only rigid bodies (no flexible bodies) without beam elements. (For more information about beam elements, refer to your Ansys Motion documentation.) Geometry component names will be retained with this type of file import. Important: Only FMU files generated by Ansys Motion will support geometry import into Rocky.	.fmu
Ansys Motion Geometry file	Used for importing geometries from an Ansys Motion file into Rocky. Currently supports only rigid bodies (no flexible bodies) without edge elements. Geometry component names will be retained with this type of file import.	.dfg
Fluent to Rocky file	File generated by the Fluent plug-in installed by Rocky. Used for importing fluid flow data out of Ansys Fluent and into Rocky. Depending upon the export type chosen, produces either a static or transient fluid flow model within the simulation. (See About Using the 1-Way Fluent Method for more information.)	.f2r

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

Type	Description	Extension
Fluent Data file	File containing Fluent initialization settings and the values of the specified flow field that can be used during a 1-Way Fluent or 2-Way Fluent coupling simulation. (See also About Using the 1-Way Fluent Method and About Using the 2-Way Fluent Method.)	.dat
Fluent Mesh Data file	File containing CFD mesh location data that can be used in a 1-Way Fluent coupling simulation. (See also About Using the 2-Way Fluent Method.)	mesh.dat
Fluent Data file (Compressed)	Compressed version of a Fluent Data file (DAT), which is an optional file containing Fluent initialization settings that are used during a 2-Way Fluent coupling simulation. (See also About Using the 2-Way Fluent Method.)	.dat.gz
Fluent Data file (HDF5)	Hierarchical Data Format (HDF) version of a Fluent data file (DAT), which is an optional file containing Fluent initialization settings that are used during a 2-Way Fluent coupling simulation. (See also About Using the 2-Way Fluent Method.)	.dat.h5
Custom Fiber definition file	Text or spreadsheet file used to define the *Segment* details that make up a Custom *Fiber* shape. (See also About Defining and Importing Custom Particle Shapes.)	.txt .csv .xls .xlsx .xlsm .xlsb .odf
Custom Input definition file	Spreadsheet file used to define the particle positioning details that make up a Custom Input. (See also About Adding and Editing Particle Inputs.)	.csv .xls .xlsx .xlsm .xlsb .odf
Point Cloud definition file	Text file used to define the details that make up a Point Cloud. (See also About Point Clouds.)	.txt

Table 2: Common Folder Locations Used in Rocky

Folder	Typical Windows Location	Typical Linux Location	Description
Rocky Settings	`%APPDATA%\Rocky`	`$HOME/.Rocky`	Folder where Rocky stores settings that persist from project to project. These include license settings, a list a list of recently opened projects, and a separate "Settings" folder that contains your display and interaction

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

Folder	Typical Windows Location	Typical Linux Location	Description
			preferences. (See below.)
Rocky Preferences	%APPDATA%\Rocky\Settings	$HOME/.Rocky/Settings	Folder where Rocky saves user preferences for various display and interaction settings. These settings can be established by users in the **Preferences** dialog (see also About Setting Global Preferences), or in several other Rocky locations by clicking a **Save Current Configuration in Settings** button. Both the values displayed on the **Preferences** dialog and the ones displayed on the UI element after the **Restore Configuration from Settings** button is clicked reflects the values saved to this folder.
User Manual Cache	%LOCALAPPDATA%\ESSS\Rocky <version>	$HOME/.local/share/ESSS/Rocky <version>	Contains the cache for the user manual.
Shared PrePost Scripts	%USERPROFILE%\Documents\Rocky\Scripts	$HOME/.Rocky/scripts	Folder where Rocky saves scripts you recorded on the **Scripts**

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

Folder	Typical Windows Location	Typical Linux Location	Description
			shared across projects tab of the **PrePost Script** panel. Also where you save PY script files that have been created outside of Rocky using the PrePost Scripting functionality. (See also Creating and Using Your Scripts.)
Project-Only PrePost Scripts	…<Project Name>rocky.files\tmp\macros-0_	…<Project Name>rocky.files/tmp/macros-0_	Folder where Rocky saves scripts you recorded on the **Project scripts** tab of the **PrePost Script** panel. Also where you save PY script files that have been created outside of Rocky using the PrePost Scripting functionality. (See also Creating and Using Your Scripts.)
External Modules	%USERPROFILE%\Documents\Rocky\Modules	$HOME/.Rocky/modules>	Folder where you save the compiled *external Module* that you either create yourself or download from the Customer Portal.

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

Tip: Performing a clean install of Rocky may involve renaming and/or archiving these folders after uninstalling Rocky and prior to reinstalling it. (See also I reinstalled Rocky but I am still seeing errors (p. 1384).)

See Also:

- Understand the User Interface (p. 74)

SIMULATION LOG FILE

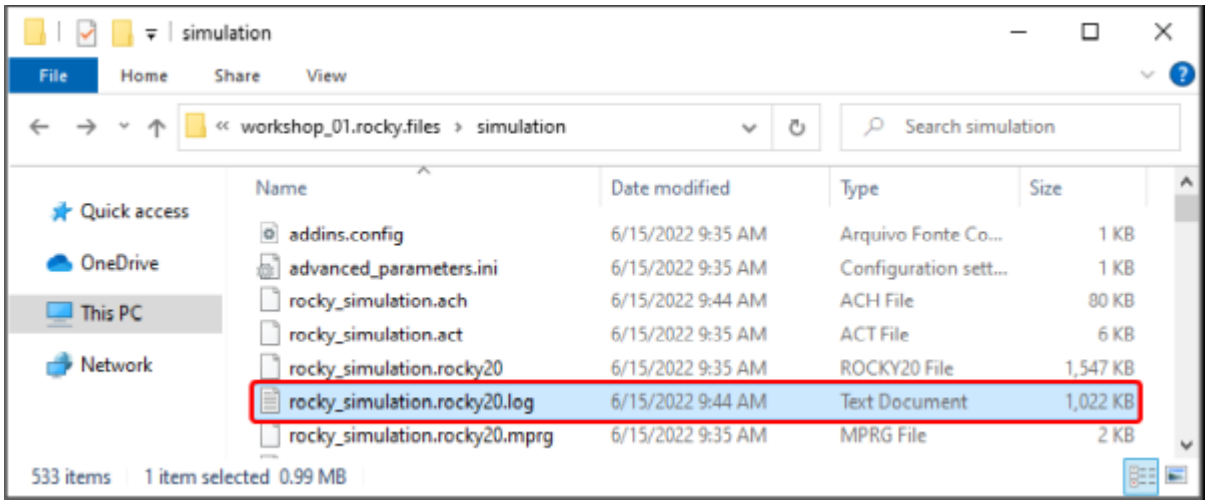
The simulation log is one of the solver output files that Rocky provides while processing a project. (See also File Types and Folders in Rocky (p. 129)). It contains simple information, such as hardware details, to more complex data about the simulation, such as particles, triangles, and contacts information; memory consumption and free memory; and other solver process information.

To access the simulation log file for a specific project, follow the steps below:

1. Navigate to the folder containing the project for which you want to see simulation details.
2. Open the **project_name.rocky.files** folder.
3. Open the **simulation** folder.

The simulation log file is located inside the simulation folder and is named **rocky_simulation.rocky20.log** (as shown in Figure 1). It is a Text Document, so you can open it with any text editor.

Figure 2.33: Example simulation log file location



FILE CONTENT

Inside the file, information is arranged similarly to the figures shown in this section. Some key areas are enumerated and explained below.

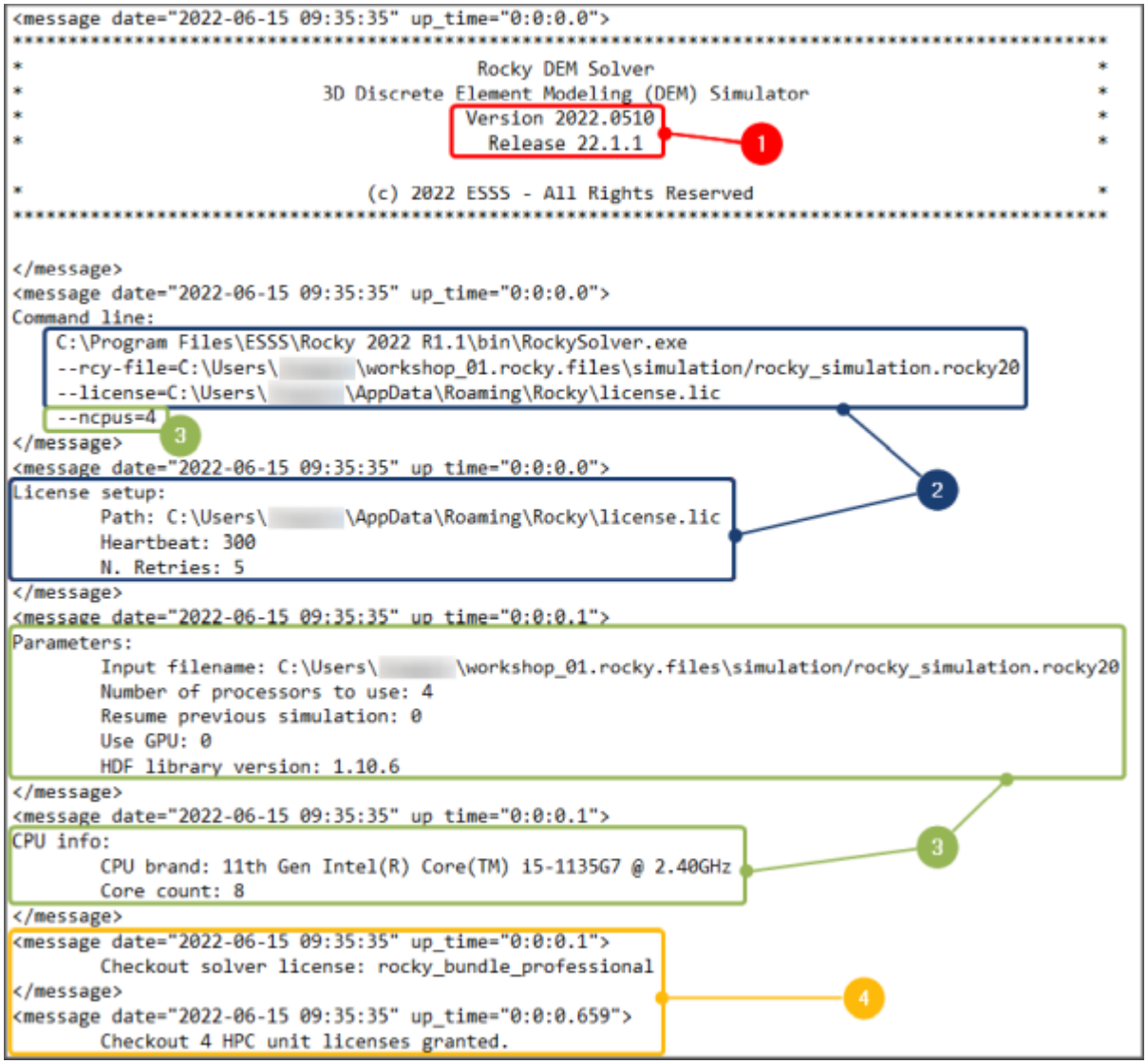
In Figure 2 (shown below), you can see the following:

1. Product release and version used to process the Rocky project. (See also About This Version of Rocky (p. 50).)

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

2. License file information
3. Hardware details
4. Used licenses. (See also Verify Your Rocky License Status (p. 1295).)

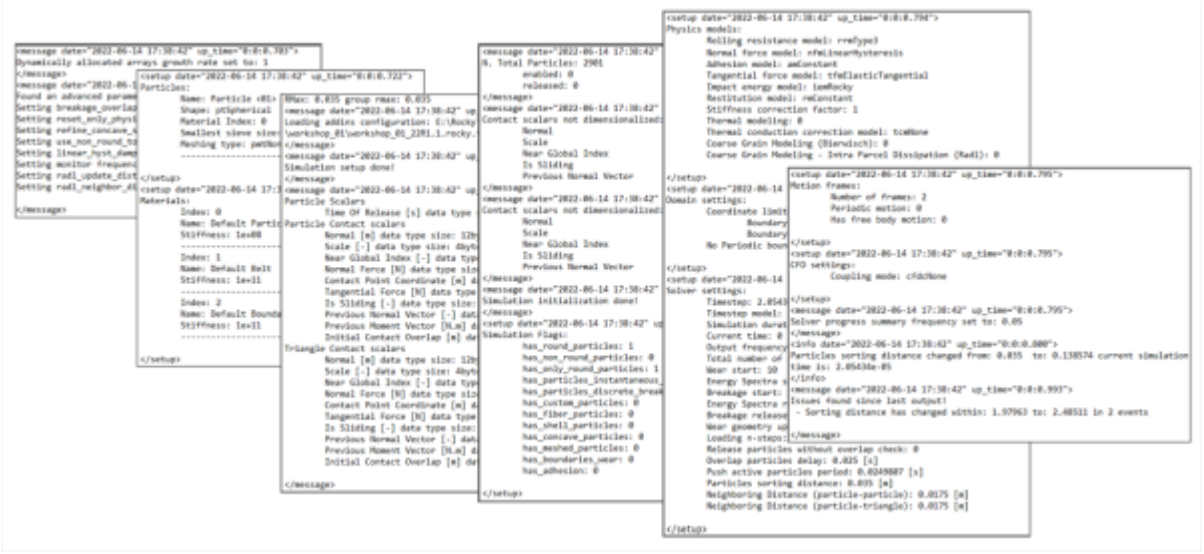
Figure 2.34: Example simulation log file showing project information



Next, you will see everything that is related to the project setup including the physical models enabled, particle groups defined, materials, boundary limits, solver options, coupling mode, motion frames, and more (Figure 3).

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

Figure 2.35: Setup information shown in an example simulation log file



After that, the file shows a structure (Figures 4 and 5) that repeats for each output time and contains the following details:

1. File number (that refers to the output time) and simulation time
2. Particle and fragment counts. (See also About Particle Breakage (p. 665).)
3. Triangle counts
4. Contact counts and ratios. (See also About Contacts (p. 746).)
5. Limits of surface velocity
6. Memory (consumption and availability). (See also About the Cache Panel (p. 103).)
7. Average time per call and number of calls for each process. **Note**: Depending on the case, the processes will be different.
8. Execution times: both the total and since the last output
9. Total execution time, which is the average time per call multiplied by the number of calls
10. Execution times for the module: both the total and since the last output
11. Total execution times, which are the total measured and segregated times by the solver in the last output plus the fraction of time not measured by the solver
12. Execution times: both the total and since the last output
13. Total execution time, which is the average time per call multiplied by the number of calls
14. Execution times for the module: both the total and since the last output
15. Total execution times, which are the total measured and segregated times by the solver in the last output plus the fraction of time not measured by the solver

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

Figure 2.36: Example simulation log file showing project data per output time

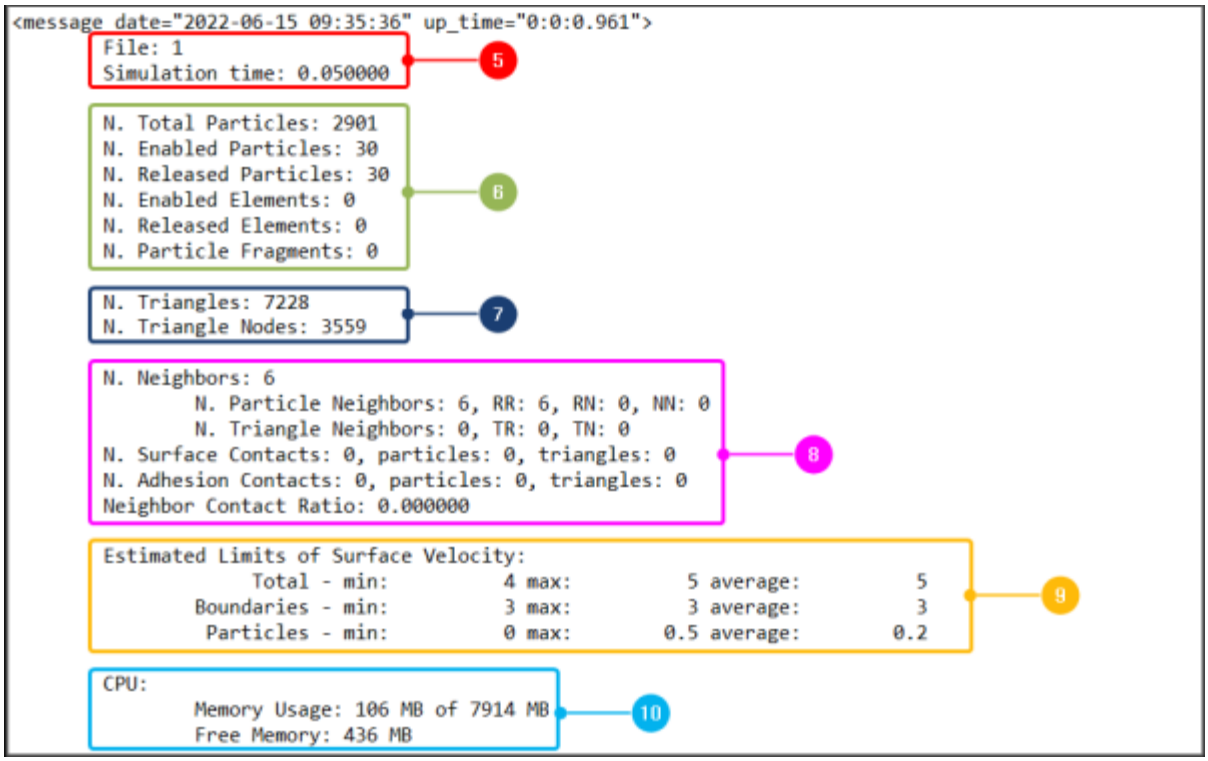
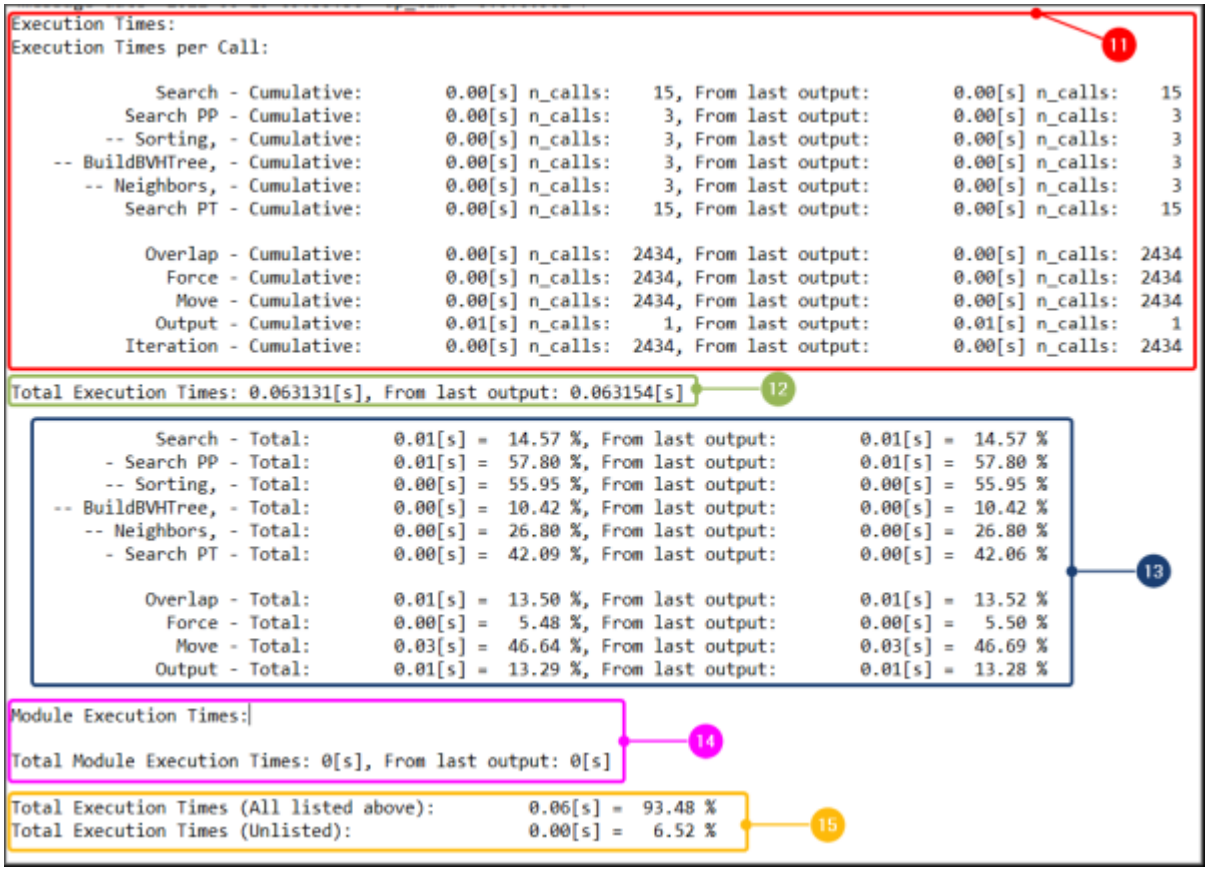


Figure 2.37: Example simulation log file showing contacts data per output time



Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

GPU-SPECIFIC INFORMATION

Finally, we have some additional information for simulations with GPU usage. Figures 6-8 show an example of a simulation log file for a project processed with 3 GPUs. The data shown includes the following:

- **GPUs IDs**

  Figure 2.38: GPUs IDs shown in the simulation log file

```
--gpu-num=0
--gpu-num=1
--gpu-num=2
--multi-gpu-slicing-direction=X
```

- **GPU load balance** with the time spent in each process and the split factor.

  Tip: The slipt factor indicates the memory consumption of the GPU in relation to the total memory consumption (from all GPUs). In this example, as we have 3 GPUs, the ideal is to have a split factor close to 0.333.

  Figure 2.39: GPU load balance status shown in the simulation log file

```
GPU load balance status: -----------------
GPU: 0 device id: 0
                time_search: 629.991
                time_force: 674.775
                time_move: 31.0968
        split_factor: 0.34566
        time_factor: 0.333625
GPU: 1 device id: 1
                time_search: 661.575
                time_force: 677.103
                time_move: 31.8351
        split_factor: 0.352662
        time_factor: 0.333749
GPU: 2 device id: 2
                time_search: 671.062
                time_force: 639.373
                time_move: 41.5631
        split_factor: 0.301678
        time_factor: 0.332626
```

- **Memory consumed** in each device

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

Figure 2.40: GPU memory status shown in the simulation log file

```
Max GPU memory usage: 152029802 = 144.987MB
Total requested memory: 151989492 = 144.948MB
--------------------------------------------------------------------------------
Memory status on device: 0
        Memory:                          cu_dev[i].adv_prm size:         168 =        0.00MB
        Memory:                              cu_dev[i].bnd size:         152 =        0.00MB
        Memory:                          cu_dev[i].bnd_mat size:         168 =        0.00MB
        Memory:               cu_dev[i].cfd_particle_groups size:         144 =        0.00MB
        Memory:                          cu_dev[i].cnt_p_0 size:     5242880 =        5.00MB
        Memory:                          cu_dev[i].cnt_p_1 size:     5242880 =        5.00MB
        Memory:                          cu_dev[i].cnt_tmp size:    78643200 =       75.00MB
        Memory:                      cu_dev[i].cnt_tot_tmp size:     3145728 =        3.00MB
        Memory:                             cu_dev[i].dims size:         168 =        0.00MB
        Memory:                          cu_dev[i].dom_prt size:          24 =        0.00MB
        Memory:                        cu_dev[i].elm_f_cfd size:          48 =        0.00MB
        Memory:                              cu_dev[i].frc size:      966768 =        0.92MB
        Memory:                              cu_dev[i].gen size:         760 =        0.00MB
        Memory:                          cu_dev[i].gen_cfd size:         280 =        0.00MB
        Memory:                           cu_dev[i].gen_lb size:         128 =        0.00MB
        Memory:                    cu_dev[i].ind_cnt_iter0 size:      524288 =        0.50MB
        Memory:                    cu_dev[i].ind_cnt_iter1 size:      524288 =        0.50MB
        Memory:                       cu_dev[i].ind_cnt_nn size:      524288 =        0.50MB
        Memory:                     cu_dev[i].ind_cnt_p_ad size:      524288 =        0.50MB
        Memory:                     cu_dev[i].ind_cnt_p_fr size:      524288 =        0.50MB
        Memory:                          cu_dev[i].ind_cst size:       80564 =        0.08MB
        Memory:                        cu_dev[i].ind_f_cfd size:           4 =        0.00MB
        Memory:                          cu_dev[i].ind_ncn size:       80564 =        0.08MB
        Memory:                          cu_dev[i].ind_per size:       80564 =        0.08MB
        Memory:                          cu_dev[i].ind_prt size:       80564 =        0.08MB
        Memory:                 cu_dev[i].ind_prt_new_f_cfd size:       80564 =        0.08MB
        Memory:                 cu_dev[i].ind_prt_old_f_cfd size:       80564 =        0.08MB
        Memory:                          cu_dev[i].ind_trn size:         640 =        0.00MB
        Memory:                          cu_dev[i].inr_prp size:          96 =        0.00MB
        Memory:             cu_dev[i].material_interactions size:         960 =        0.00MB
        Memory:                        cu_dev[i].materials size:         200 =        0.00MB
```

- **N. Halo SPH Elements/Halo Particles:** In *multi-GPU* runs, halo particles/elements are those that are located in the overlapping devices domain regions and therefore are present simultaneously in two GPU devices. For the halo particles/elements, both devices compute the collision forces, but only one of the devices accumulates these data in order to perform velocity and displacement calculations.

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

Figure 2.41: N. Halo SPH Elements and N. Enabled Coupled SPH Elements

```
SPH:
      N. Total SPH Elements: 3822
      N. Enabled SPH Elements: 3822
      N. Total Coupled SPH Elements: 0
      N. Enabled Coupled SPH Elements: 0
      N. Halo SPH Elements: 196
      N. Halo SPH Elements Ratio: 5.13%
```

- **Problem Partitioning Status:** This parameter represents the number of SPH Elements or particles coupled in the simulation.

Figure 2.42: Problem Partitioning Status in a DEM-only simulation

```
Problem Partitioning Status: ----------------------------------------------------

      GPU: 0 id(0):
            N. Enabled Particles: 2405 (49.99% of Total)
            N. Halo Particles: 0
            N. Neighbors: 14475
                  N. Particle Neighbors: 13632, RR: 13632, RN: 0, NN: 0
                  N. Triangle Neighbors: 843, TR: 843, TN: 0
            N. Surface Contacts: 0, particles: 0, triangles: 0
            N. Adhesion Contacts: 0, particles: 0, triangles: 0
            Neighbor Contact Ratio: 0.000000

      GPU: 1 id(1):
            N. Enabled Particles: 2406 (50.01% of Total)
            N. Halo Particles: 663
            N. Neighbors: 12406
                  N. Particle Neighbors: 11604, RR: 11604, RN: 0, NN: 0
                  N. Triangle Neighbors: 802, TR: 802, TN: 0
            N. Surface Contacts: 0, particles: 0, triangles: 0
            N. Adhesion Contacts: 0, particles: 0, triangles: 0
            Neighbor Contact Ratio: 0.000000
```

- **N. Enabled Coupled SPH Elements:** It is the number of SPH elements figured inside the particles for the purpose of interacting with external SPH elements in an SPH-DEM interaction simulation. This parameter will only be computed in simulations with SPH and DEM, otherwise it will appear as zero.

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

Figure 2.43: Problem Partitioning Status in a SPH-only simulation

```
Problem Partitioning Status: ----------------------------------------------------

      GPU: 0 id(1):
            N. Enabled SPH Elements: 1911 (50.00% of Total)
            N. Enabled Coupled SPH Elements: 0
            N. Halo SPH Elements: 0

      GPU: 1 id(2):
            N. Enabled SPH Elements: 1911 (50.00% of Total)
            N. Enabled Coupled SPH Elements: 0
            N. Halo SPH Elements: 196
```

Note:

If you processed your simulation without GPU usage, your simulation log file will not show this data. You can see how many GPUs were used by looking at the section marked "3" in Figure 1. (In that simulation example, "Use GPU: 0" means that the project was processed without GPU usage.)

2.3. Ansys Rocky Coupling with other Ansys Software

Rocky comes integrated with several Ansys products including Mechanical, Fluent, Motion, Minerva, and optiSLang.

Rocky can also integrate with Ansys Workbench. Working through Workbench enables Rocky to couple its abilities with other Ansys products - including Mechanical, Fluent, DesignXplorer, and SpaceClaim - which enables you to define design points for parametric cases, and solve more kinds of solutions in more complex and innovative ways.

When a Rocky project is connected to Ansys Mechanical though Workbench, an **External Coupling** component becomes enabled in the Rocky **Data** panel. This component enables you to define the kind of data gets shared between the programs, such as the geometry loads.

This section covers the following topics:

- Ansys Rocky and Ansys Workbench (p. 144)
- Ansys Rocky and Ansys Mechanical (p. 170)
- Ansys Rocky and Ansys Fluent (p. 197)
- Ansys Rocky and Ansys Motion (p. 208)
- Ansys Rocky and Ansys optiSLang (p. 209)
- Ansys Rocky and Ansys Minerva (p. 217)

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

2.3.1. Ansys Rocky and Ansys Workbench

Ansys Workbench is a product you can use to more easily connect, share, automate, and keep organized data across various programs. When used with Rocky, Workbench can help you accomplish many time-saving tasks like the ones described in the sections below.

Tip: To learn which versions of Ansys-including Workbench, SpaceClaim, Mechanical, Fluid, and other products-are compatible with this version of Rocky, refer to System Requirements (p. 67).

This section covers the following topics:

ROCKY INTEGRATION IN WORKBENCH (p. 144)

WORKBENCH INTEGRATION IN ROCKY (p. 146)

MAXIMIZE THE DESIGN OF YOUR GEOMETRY COMPONENTS BY USING ANSYS SPACECLAIM (p. 148)

PARAMETERIZE SIMULATION AND SETUP COMPONENTS BY USING ANSYS DESIGNXPLORER (p. 149)

CONDUCT FEA ANALYSIS OF PARTICLE FORCES ACTING UPON GEOMETRY COMPONENTS BY USING ANSYS MECHANICAL (p. 149)

STUDY HOW CFD FLUID AND THERMAL PROPERTIES AFFECT PARTICLE FLOW BY USING ANSYS FLUENT (p. 150)

1-WAY AND 2-WAY COUPLING WITH FLUENT IN WORKBENCH (p. 151)

1-WAY ANSYS FLUENT COUPLING WITH WORKBENCH (p. 152)

UNRESOLVED 2-WAY ANSYS FLUENT COUPLING WITH WORKBENCH (p. 158)

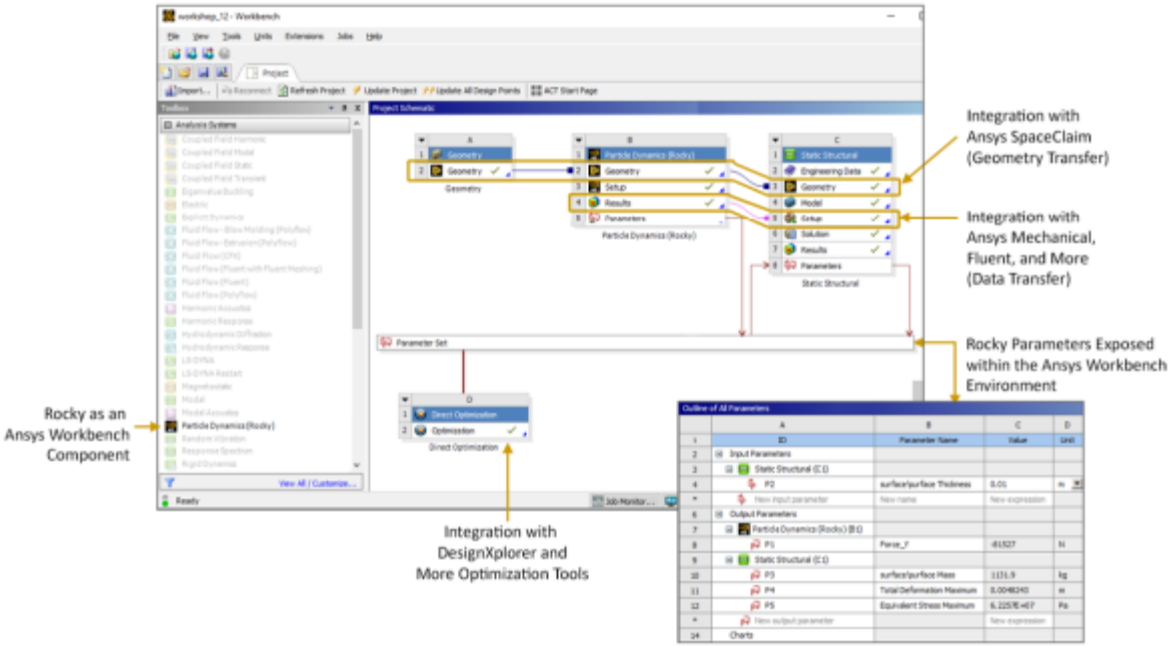
SEMI-RESOLVED 2-WAY ANSYS FLUENT COUPLING WITH WORKBENCH (p. 167)

ROCKY INTEGRATION IN WORKBENCH

Once you open Workbench and create a new Project, you can from the **Toolbox** panel, expand the **Analysis Systems** item, and then drag and drop **Particle Dynamics (Rocky)** to the **Project Schematic** (Figure 8). Note: This item will be referred to as the "Rocky" block for simplicity.

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

Figure 2.44: Particle Dynamics (Rocky) as an Ansys Workbench component

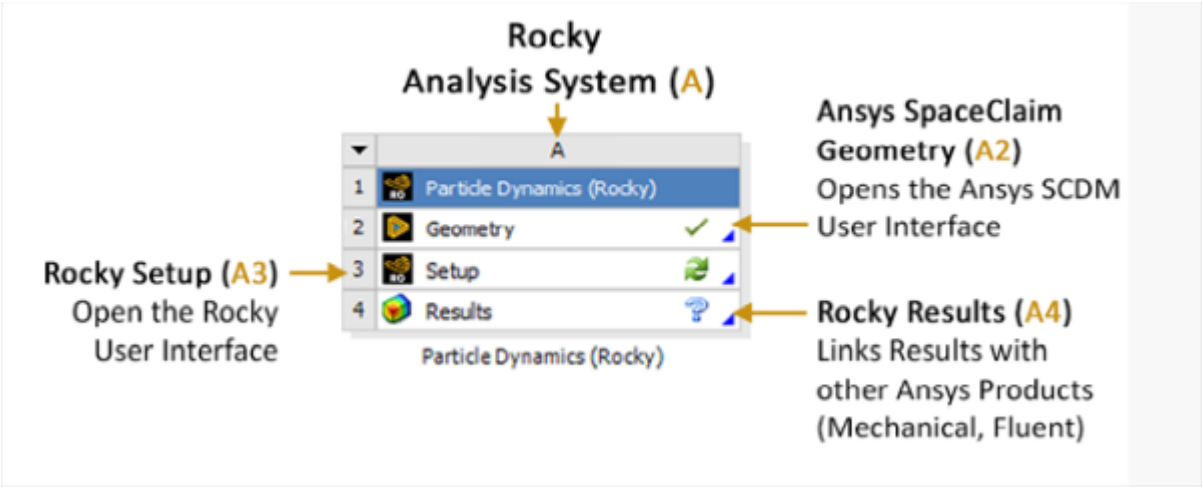


This creates a link between your Workbench project and the Rocky program. From here, you can either start a new Rocky project from within Workbench (Edit), or import a Rocky project into Workbench that you have already set up (Import Setup).

The link between your Workbench project and your Rocky project means that data will be automatically transferred between the two programs. In addition, it exposes Rocky input and output parameters, so you can explore Workbench capabilities such as evaluating multiple design points, creating Rocky cases automatically, and then running them through the weekend. (See also About Defining and Using Input Variables (p. 243) and About Defining Output Variables (p. 250).)

The new **Rocky** Analysis System block (A) is made up of the components described in Figure 9.

Figure 2.45: Rocky Analysis System block in Workbench



By interacting with the **Rocky** Analysis System block, and linking other items from the Workbench Analysis System-such as the Transient Structural block or the Static Structural block-to the Rocky items

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

listed, you can have Workbench perform many different tasks in Rocky. This includes one-way coupled approaches with Ansys Mechanical, one-way coupled approaches with Ansys Fluent, standalone parametric studies, and more.

Note:

In this version of Rocky, projects coupled with Ansys Workbench now use Ansys Discovery as default for Geometry imports and conversions. Therefore, if Ansys Discovery is not installed, you will get an error message when trying to convert geometries into .STL in Rocky-Workbench coupled projects.

Important:

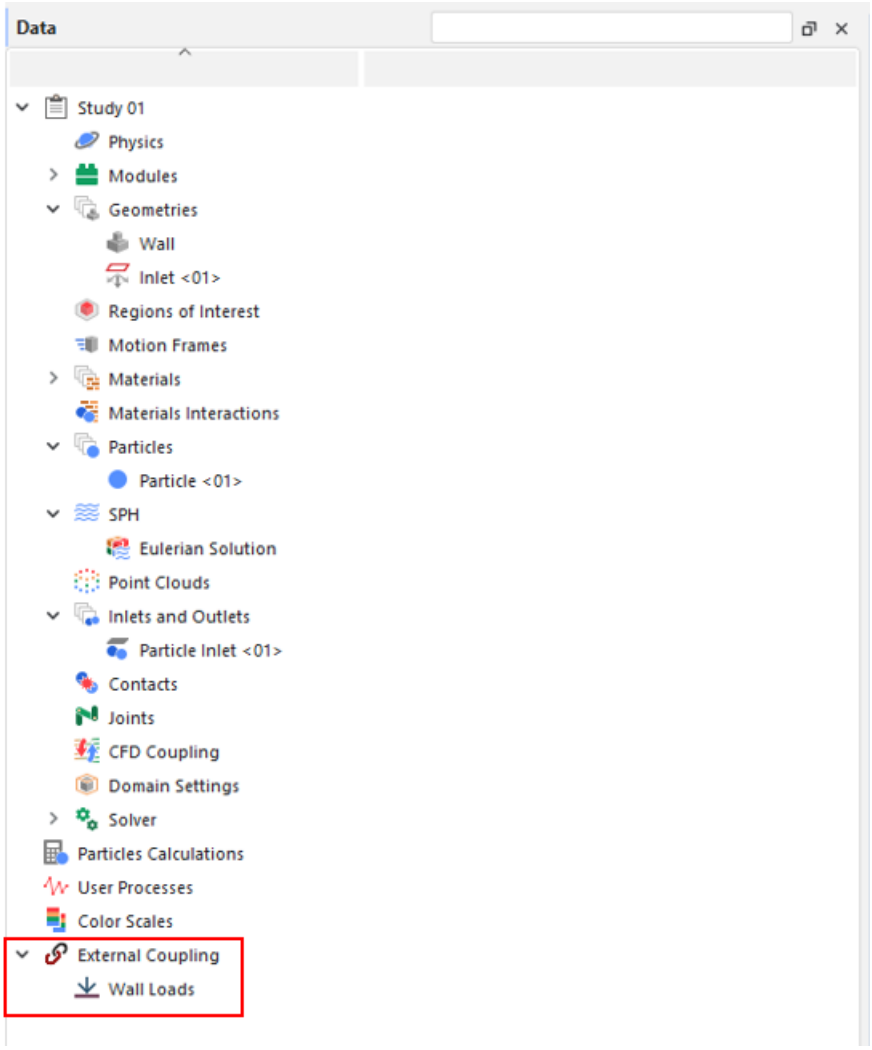
Prime is used as the STL exporting tool from Discovery software, in this way, even if the components are placed in hidden or invisible mode they will still be exported to Rocky.

WORKBENCH INTEGRATION IN ROCKY

If you have linked to Rocky from within Workbench, an **External Coupling** entry will appear in the Rocky **Data** panel (Figure 10).

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

Figure 2.46: Rocky connected to Mechanical via Workbench, showing a new External Coupling entry



For Rocky projects opened through Workbench that are not coupled with Ansys Mechanical, this component will still appear in the **Data** panel but can be ignored.

WORKBENCH INTEGRATION RESOURCES

To learn more about Workbench integration with Rocky, and to see Workbench in action, refer to the following resources:

- Tutorial - Static Structural in the *Rocky Tutorial Guide*
- Tutorial - Transient Structural in the *Rocky Tutorial Guide*

See Also:

- About Rocky and Ansys Mechanical Integration (p. 170)
- About Rocky and Ansys Fluent Integration (p. 197)
- About Defining and Using Input Variables (p. 243)

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

- About Defining Output Variables (p. 250)
- About Defining External Coupling Options (p. 839)
- Install Ansys Coupling Components (p. 792)
- I cannot find the Rocky component in Workbench (p. 1327)

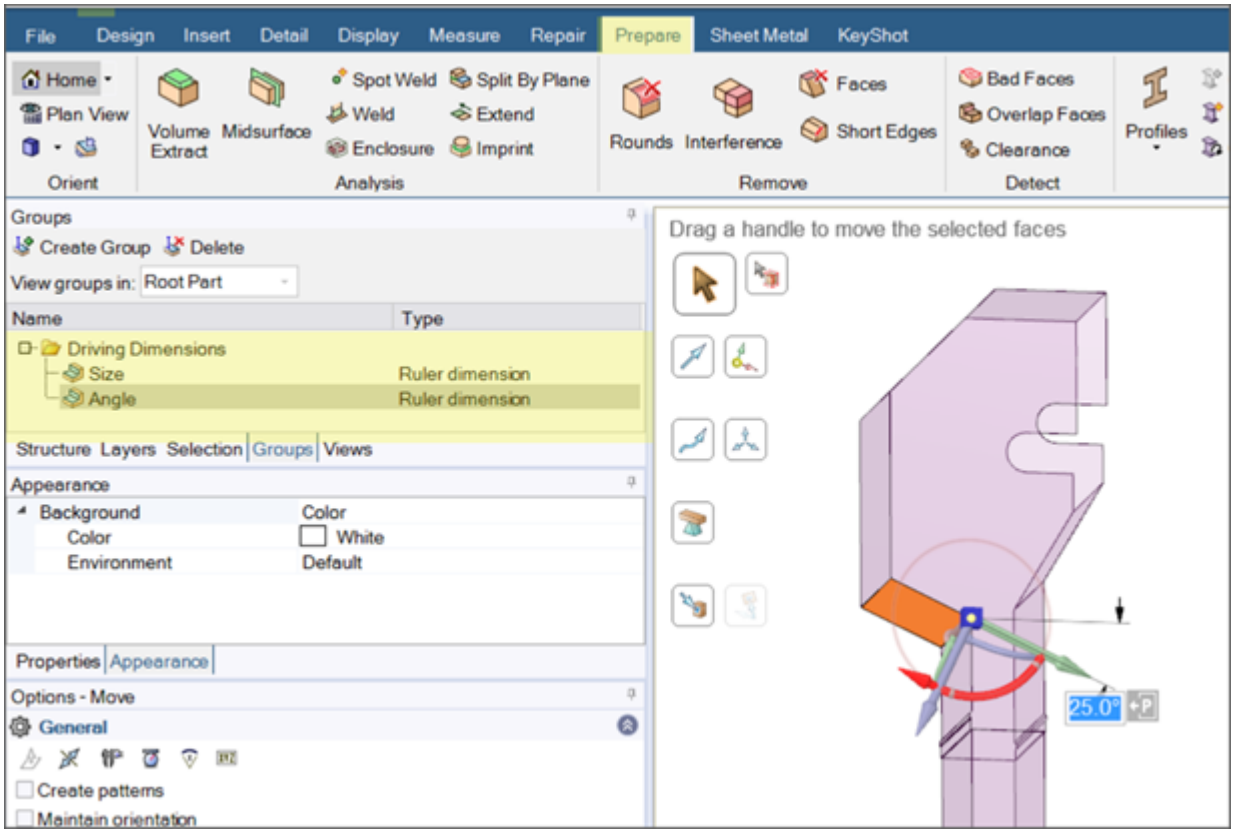
MAXIMIZE THE DESIGN OF YOUR GEOMETRY COMPONENTS BY USING ANSYS SPACECLAIM

This is accomplished through Workbench by connecting your Rocky project with Ansys SpaceClaim 3D geometry files, thereby enabling you to change equipment components in SpaceClaim (Figure 1) and have those geometry changes be automatically updated in your Rocky project.

Note:

In this version of Rocky, projects coupled with Ansys Workbench now use Ansys Discovery as default for Geometry imports and conversions. Therefore, if Ansys Discovery is not installed, you will get an error message when trying to convert geometries into .STL in Rocky-Workbench coupled projects.

Figure 2.47: Geometry component being modified in SpaceClaim



Tip: To see walk-through examples of modifying your geometries in SpaceClaim for use in your Workbench-connected Rocky projects, refer to the following Tutorials:

- Tutorial - Static Structural in the *Rocky Tutorial Guide*

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

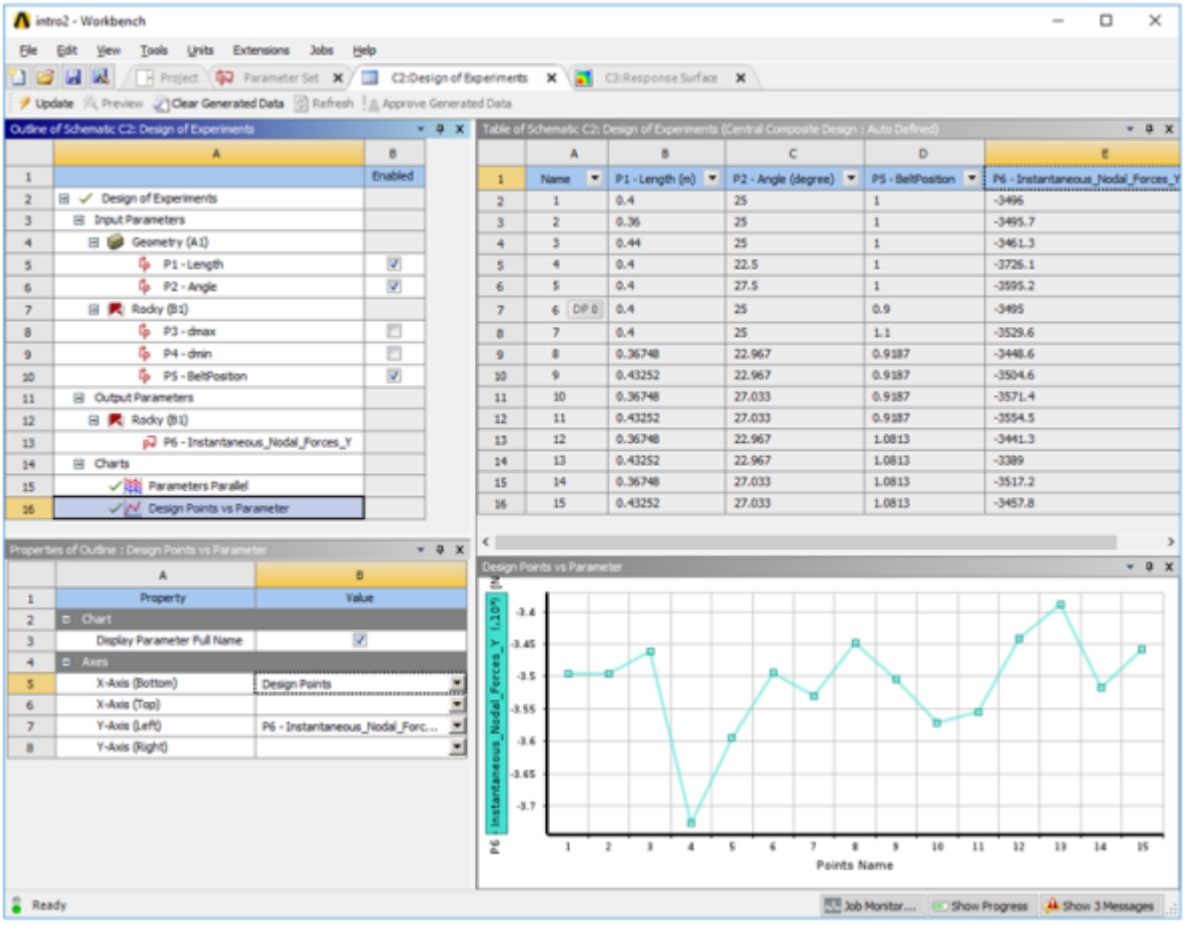
- Tutorial - Windshifter in the *Rocky Tutorial Guide*
- Tutorial - Transient Structural in the *Rocky Tutorial Guide*

PARAMETERIZE SIMULATION AND SETUP COMPONENTS BY USING ANSYS DESIGNXPLORER

This is accomplished through Workbench by connecting your Rocky project with Ansys DesignXplorer, which enables you to set design goals and get immediate feedback on proposed changes. DesignXplorer includes correlation, design of experiments, response surface creation and analysis, optimization and six sigma analysis. In addition, integration with other Ansys products like SpaceClaim ensures that any geometry changes made are automatically updated in your Rocky project.

An example of such a project is shown in Figure 2 where varied geometrical inputs as well as Rocky inputs are used to evaluate the impact of those changes in Rocky output parameters. (See also About Defining and Using Input Variables (p. 243) and About Defining Output Variables (p. 250).)

Figure 2.48: Example of a parametrized Rocky project using DesignXplorer



Using DesignXplorer through Workbench can be especially helpful when testing how small changes in geometry angles, sizes, or placements affects your material flow.

CONDUCT FEA ANALYSIS OF PARTICLE FORCES ACTING UPON GEOMETRY COMPONENTS BY USING ANSYS MECHANICAL

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

By using Workbench to connect your Rocky project with Ansys Mechanical, you can have Mechanical predict the stress and strain response of your geometry components when they are subjected to the granular loads that are calculated in Rocky.

Workbench does this by first having Rocky calculate the particle forces and pressures on the geometry component. Next, it automatically exports that data from Rocky to Mechanical through Workbench, where Mechanical then uses those forces and pressures to perform static (at a single time - Figure 3) or transient (over a time range - Figure 4) structural analysis on the component.

Figure 2.49: Example of static structural FEA analysis

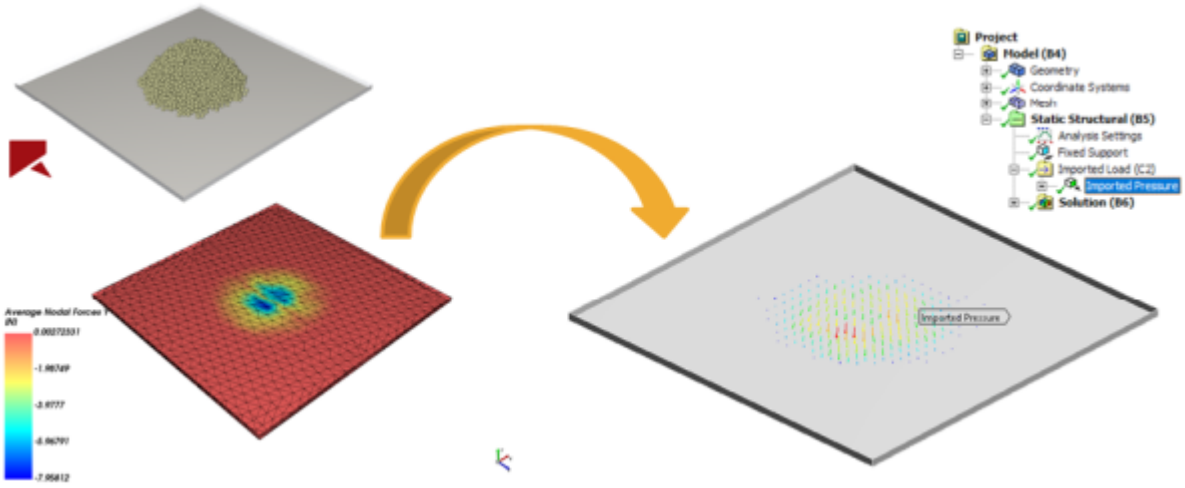
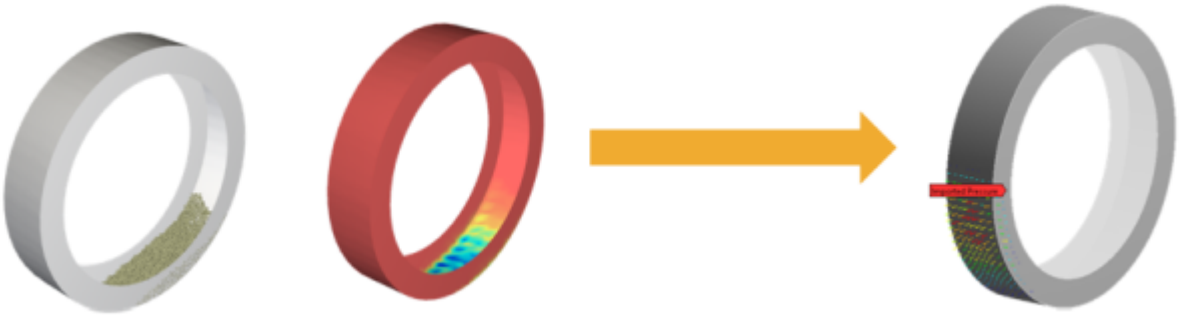


Figure 2.50: Example of a transient structural FEA analysis



This approach is useful for analyzing the stress effects of particles upon a vibrating screen, for example, or for visualizing the mechanical deformation of a bucket excavator due to particle loads.

Tip: To learn more about using Mechanical to conduct Static and Transient FEA analyses in your Workbench-connected Rocky projects, refer to the following resources:

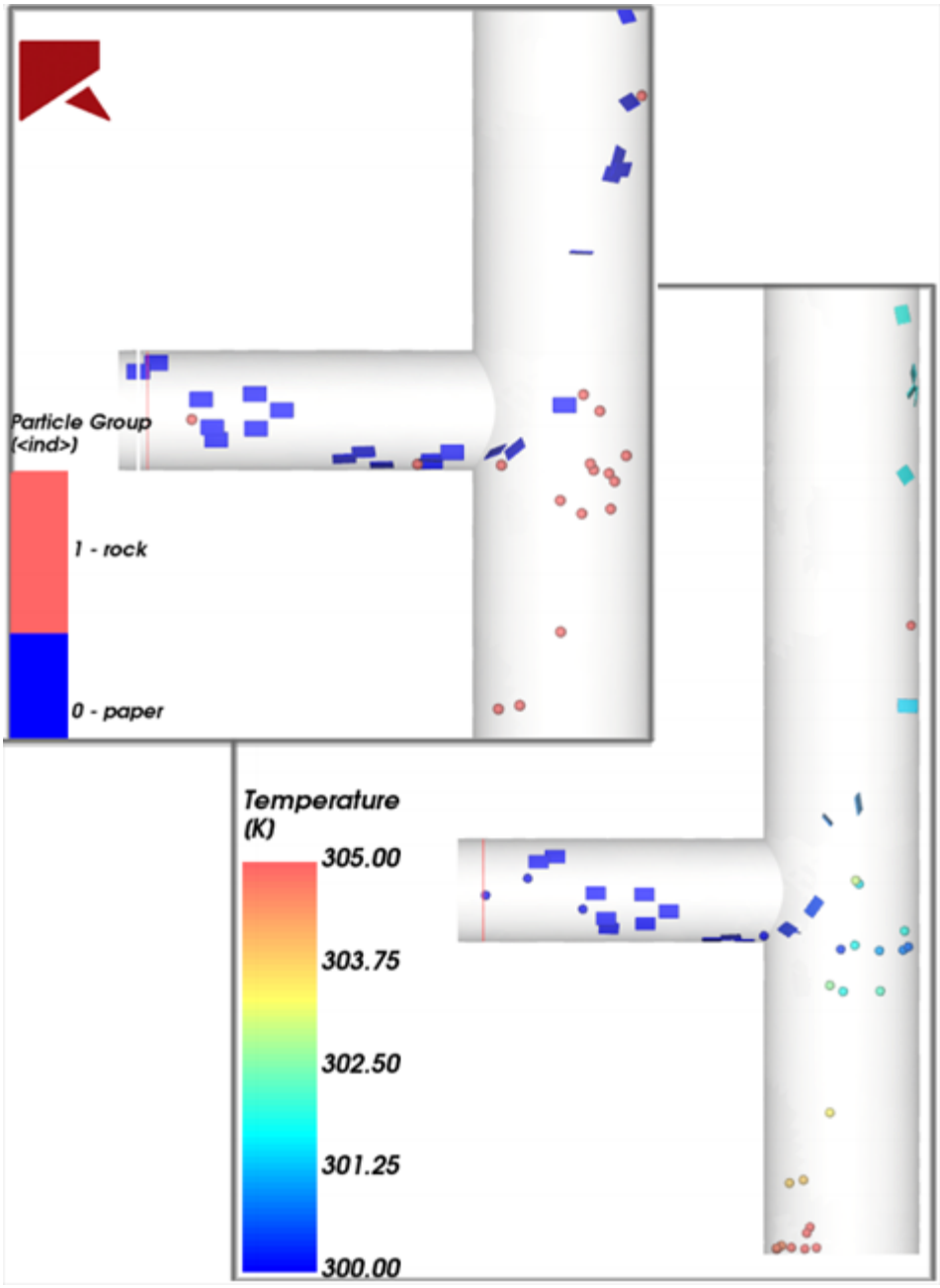
- Tutorial - Static Structural in the *Rocky Tutorial Guide*
- Tutorial - Transient Structural in the *Rocky Tutorial Guide*
- Tutorial - Backhoe Loader in the *Rocky Tutorial Guide*

STUDY HOW CFD FLUID AND THERMAL PROPERTIES AFFECT PARTICLE FLOW BY USING ANSYS FLUENT

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

By using Workbench to couple your Rocky project 1-Way with Ansys Fluent, Rocky can calculate how the CFD fluid flow and heat transfer affects the DEM particle flow and related thermal properties (Figure 5).

Figure 2.51: Example of a 1-Way Fluent Coupled Simulation with Rocky



This approach is useful for studying how air flow affects various kinds of material flowing through a pipe, for example.

Tip: To see a walk-through example of a 1-Way coupled simulation with Ansys Fluent, refer to the following Tutorial: Tutorial 13 - DEM-CFD One Way Coupling With Ansys Fluent (Workbench).

1-WAY AND 2-WAY COUPLING WITH FLUENT IN WORKBENCH

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

From Rocky 23R1 it's possible to use both 1-way and 2-way CFD-DEM coupling analyses through Ansys Workbench. This functionality brings many advantages, such as:

- **Multiphysics Analyses Integration:** With a single interface, it's possible to integrate multiple analyses, making it easy to see in the workflow project who is sending data to whom.
- **Automated Data Management:** Save time without the need to manually transfer the results from one application to another.
- **Run multiple designs:** to explore different scenarios.
- **Optimizing Exploration:** parametric modeling capabilities in conjunction with optimization techniques to allow you to efficiently investigate the effects of input parameters on selected output parameters.
- **Workflow:** all the necessary files are archived together, making it easy to share the project with other users.

To couple Rocky and Fluent in Workbench, follow the steps below:

1-WAY ANSYS FLUENT COUPLING WITH WORKBENCH

1. The 1-way coupling can use either a steady or transient flow field. For the 1-way coupling, select Fluent Solution Properties and change the **Initialization Method** to **Solver Controlled**;

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

Figure 2.52: For 1-way coupled simulations, the Initialization Method should be set as "Solver Controlled" in the Fluid Flow Solution Properties.

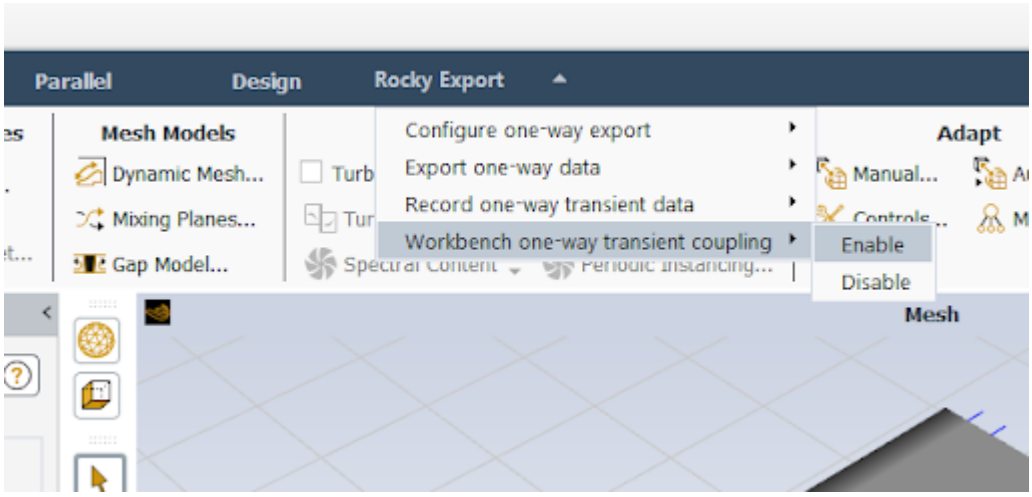
Properties of Schematic C5: Solution

	A	B
1	Property	Value
2	General	
3	Component ID	Solution 3
4	Directory Name	FFF-3
5	Use Setup Launcher Settings	☑
6	Precision	Double Precision
7	Show Launcher at Startup	☑
8	Display Mesh After Reading	☑
9	Embed Graphics Windows	☑
10	Use Workbench Color Scheme	☑
11	Load ACT Start Page	☐
12	Environment Path	
13	Setup Compilation Environment for UDF	☑
14	Use Job Scheduler	☐
15	Run Parallel Version	☑
16	UDF Compilation Script Path	$(FLUENT_ROOT)\$(ARCH)\udf.bat
17	Initialization Method	Solver Controlled
18	Use Remote Linux Nodes	☐
19	Solution Monitoring	☑
20	Generate Solution Monitor Plots for Report	☐
21	Generate Data Interpolation File	☑
22	Generate Post Processing Images	☑
23	Notes	
24	Notes	
25	Used Licenses	
26	Last Update Used Licenses	CFD Base CFD Solver - Level 1 CFD Solver - Level 2
27	Parallel Run Settings	
28	Number of Processors	4
29	Interconnect	default
30	MPI Type	default
31	Use Shared Memory	☑
32	Solution Process	
33	Update Option	Run in Foreground

2. For a **1-way transient coupling**, the option to **Export the transient data** must be enabled in the **Rocky Export entry in Fluent**.

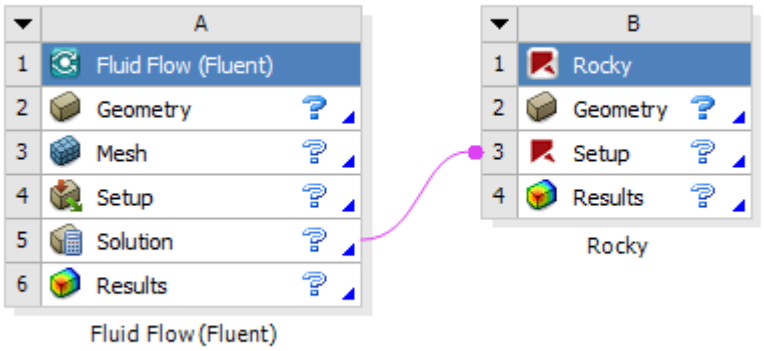
Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

Figure 2.53: Enable export the transient data.



3. To use the flow field in the 1-way coupled simulation, link the **Fluent Solution** to the **Rocky Setup**.

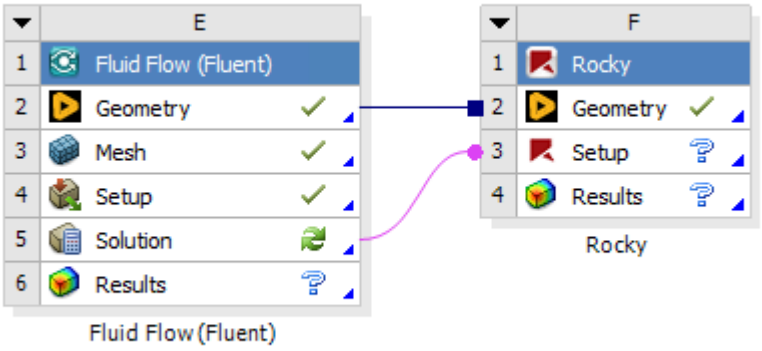
Figure 2.54: Fluent-Rocky 1-way coupling.



4. For sharing the geometry between projects, link the **Geometry** of the Fluid project to the **Geometry** of the Rocky project. Doing so, the geometry will be automatically imported into Rocky.

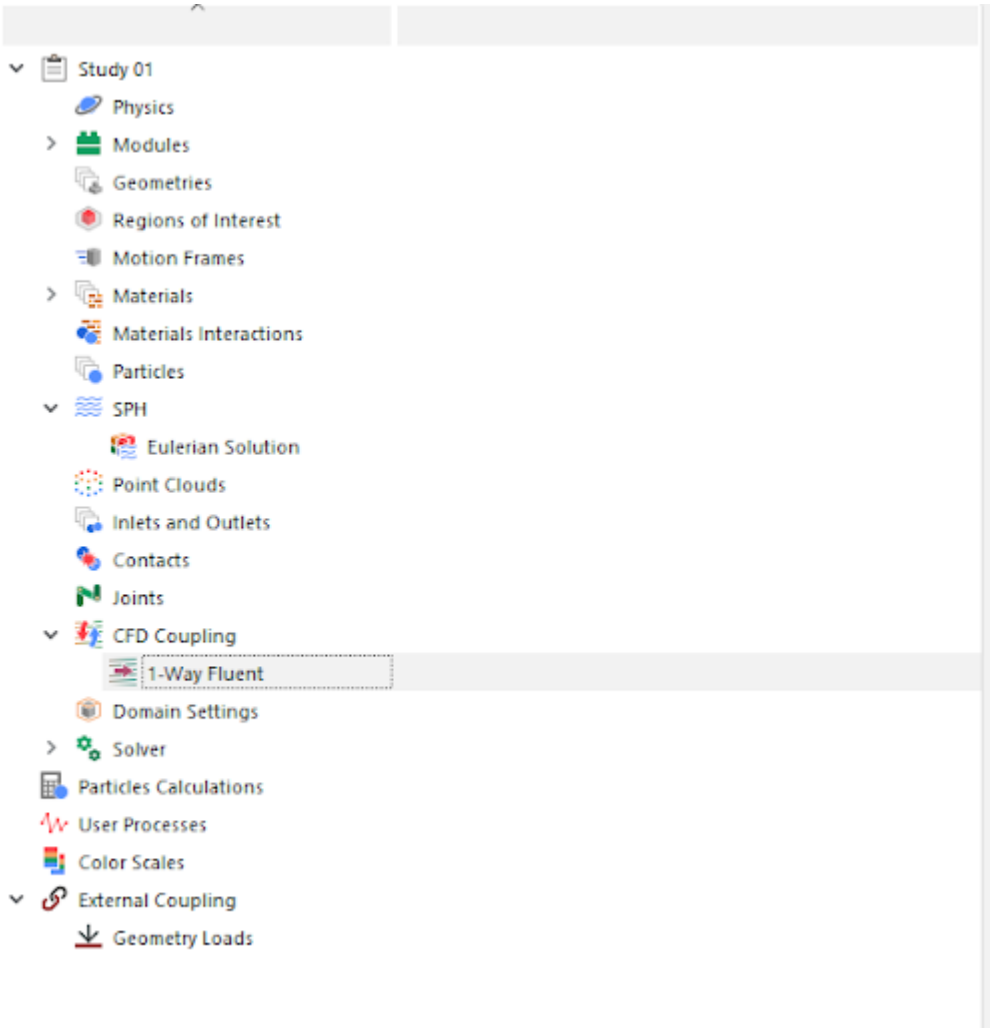
Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

Figure 2.55: Sharing the geometry between projects.



5. After the CFD solution is ready, when opening Rocky, the CFD Coupling entry in the Data Tree has the 1-Way Fluent mode selected.

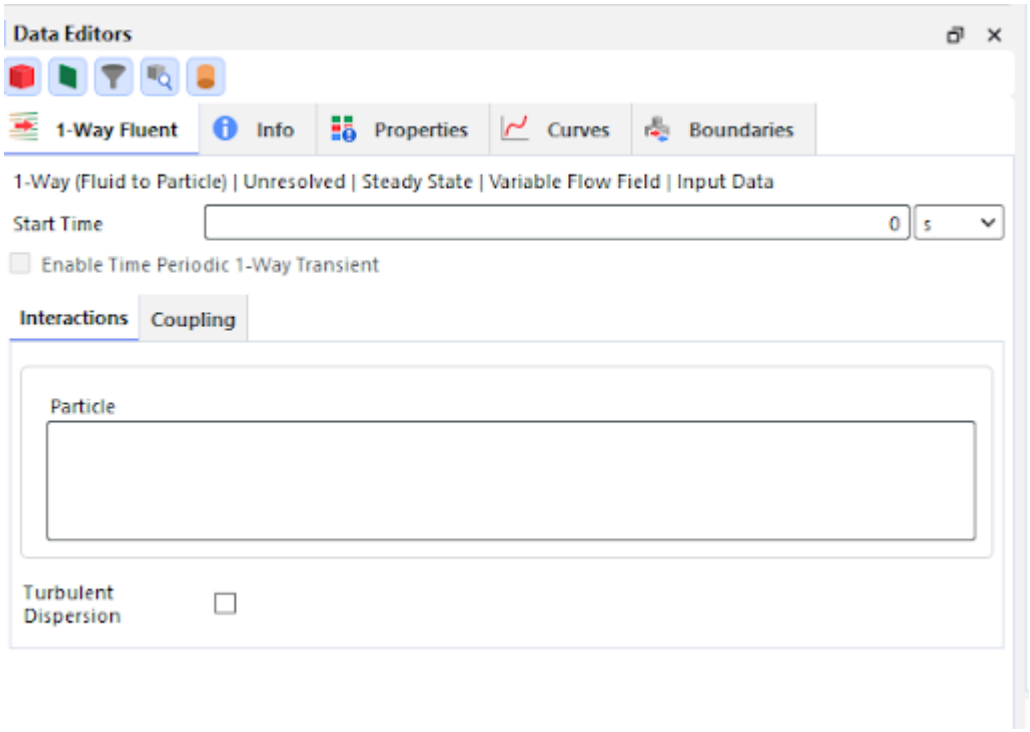
Figure 2.56: 1-Way Fluent mode selected in Data Tree.



Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

6. Open the 1-Way Fluent Data Editor and edit the coupling settings as usual.

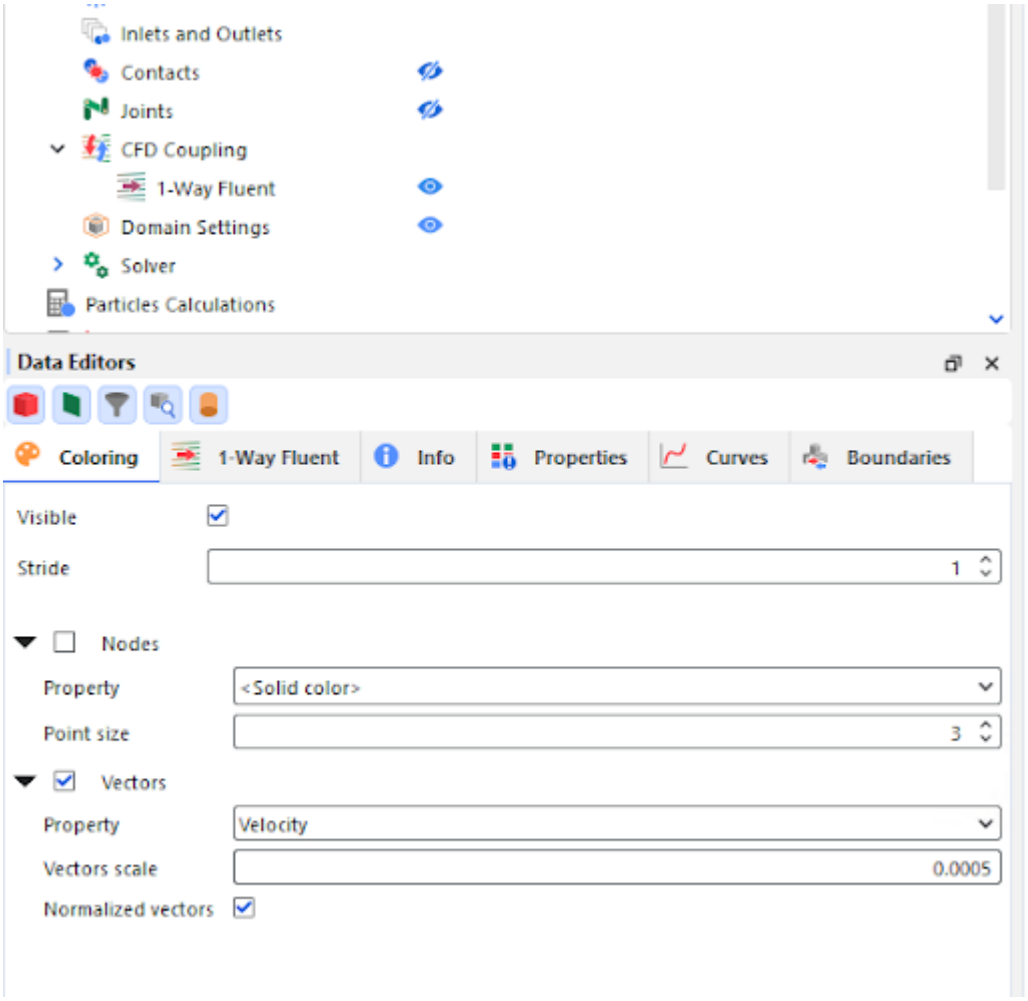
Figure 2.57: Data Editor for the 1-way steady-state coupling between Rocky and Fluent.



7. After run the simulation, the CFD and DEM results of the unresolved 1-way coupling can be simultaneously post-processed in Rocky Fluid. Quantities such as pressure and velocities can be analyzed and shown in the 3D view together with the particles.

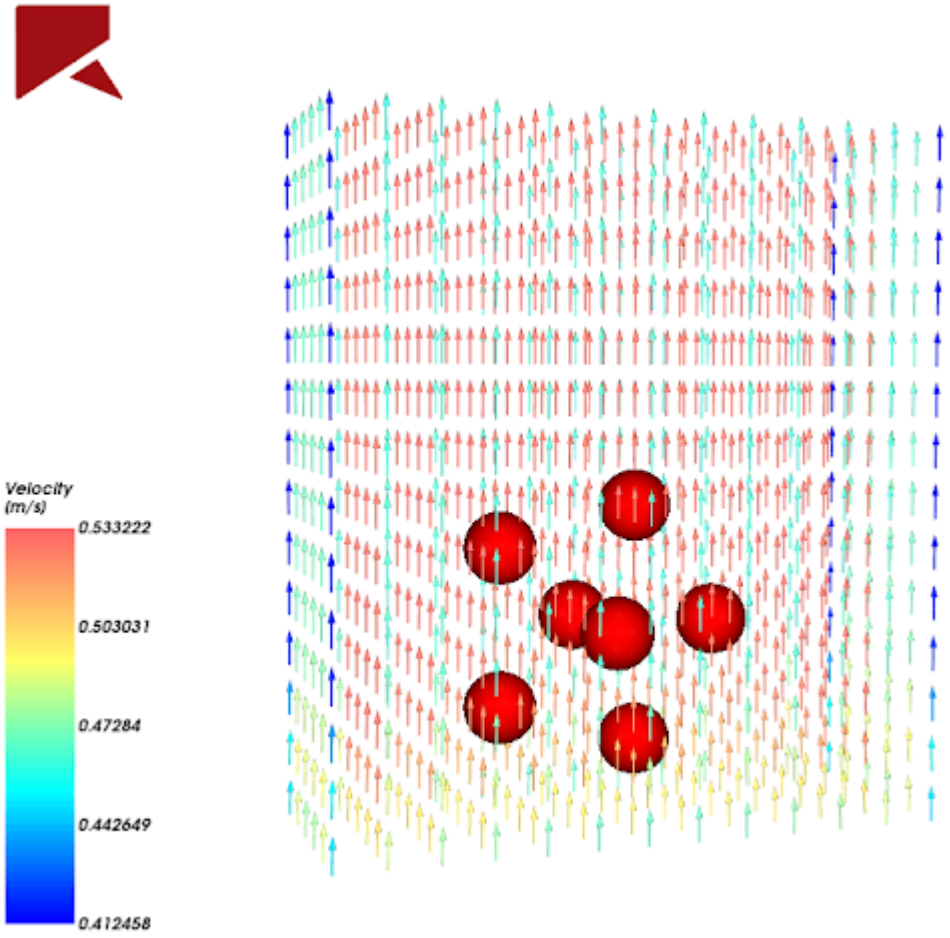
Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

Figure 2.58: The coloring tab of the 1-Way Fluent Data Editor allows the selection of which fluid information should be showed in the 3D view.



Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

Figure 2.59: Vectors colored by the magnitude of the fluid velocity give insight to the flow field in 1-way coupled simulations.



UNRESOLVED 2-WAY ANSYS FLUENT COUPLING WITH WORKBENCH

1. Drag the Fluent analysis onto the Project Schematic and set up the Fluent Project.

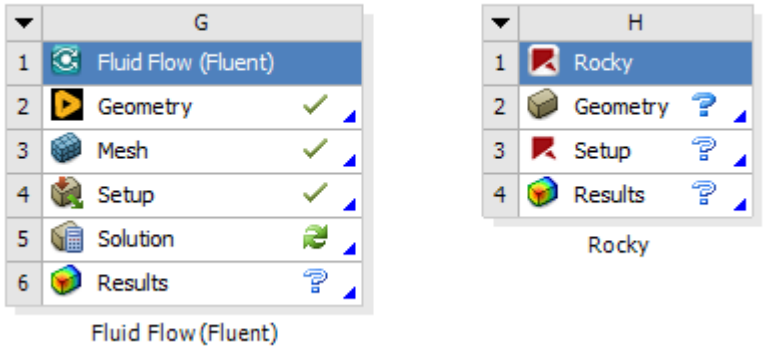
   Important: You can only set up the 2-way coupled project if the Fluent Project is set up. Rocky looks for the .cas file for importing the necessary files for the 2-way coupling.

2. Drag a Rocky Analysis to the Project Schematics. Do not drop it on top of the Fluent setup.

   Important: Do not drop it on top of the Fluent setup. The link must be created manually after both the two systems are placed at the Project Schematic.

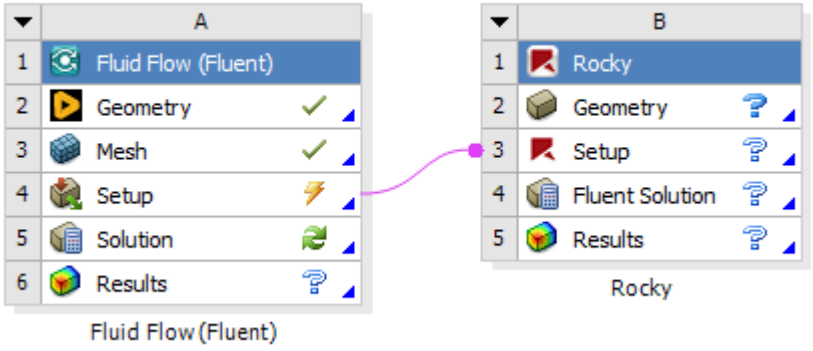
Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

Figure 2.60: Draging a Rocky Analysis to the Project Schematics.



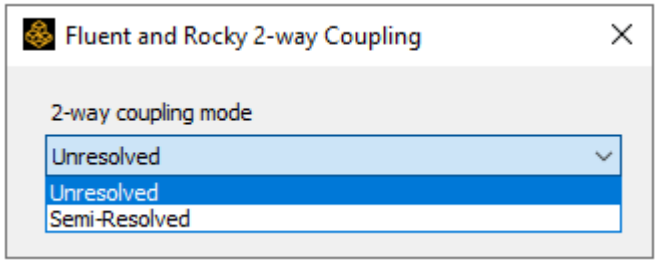
3. Drag the **Fluent Setup** onto the **Rocky Setup**.

Figure 2.61: Fluent-Rocky 2-way coupling.



4. Select **Unresolved** as the 2-way coupling mode.

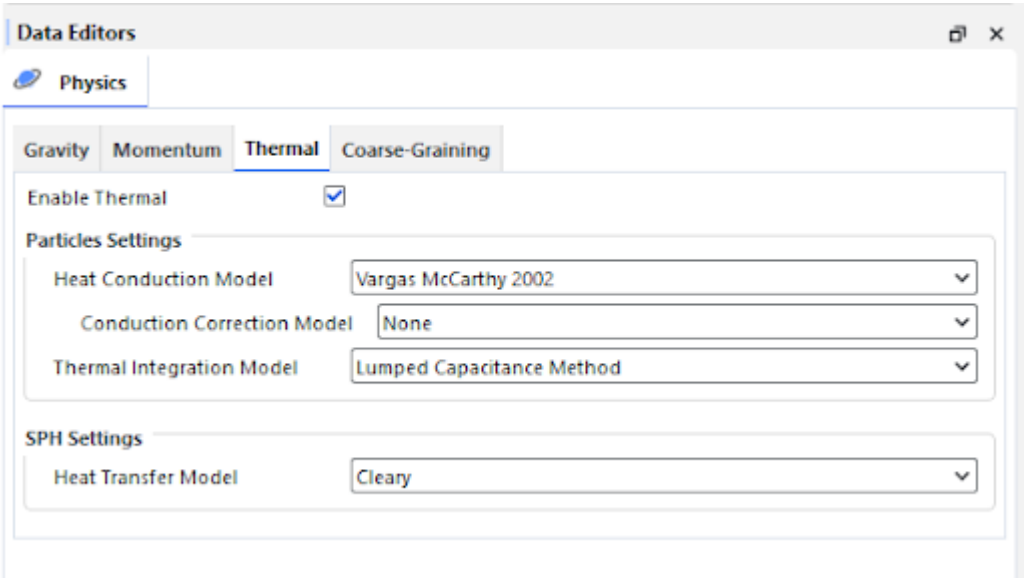
Figure 2.62: Unresolved 2-way coupling mode.



5. If the Fluent setup has Energy Model on, enable the Thermal in the Momentum tab of the Physics Data Editor in Rocky.`

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

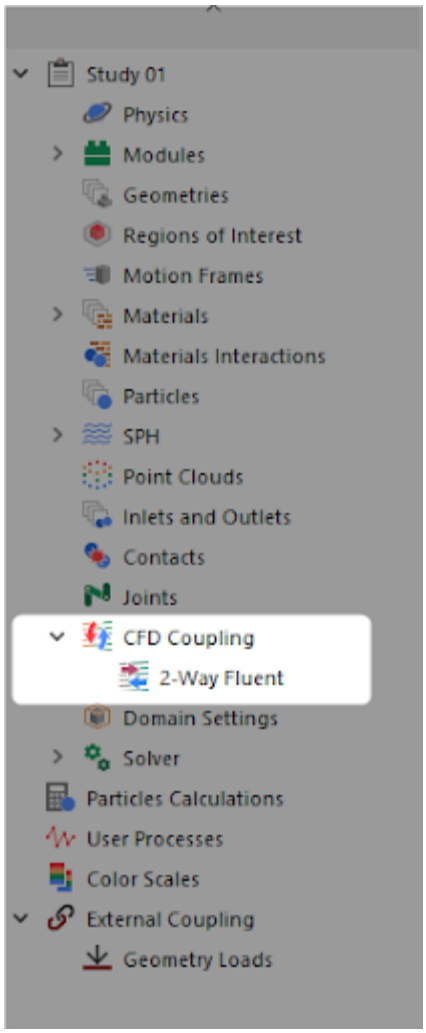
Figure 2.63: Enabling the Thermal in the Momentum tab of the Physics Data Editor in Rocky.



6. When opening Rocky, the CFD Coupling entry at the Data Tree has the 2-Way Fluent mode selected.

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

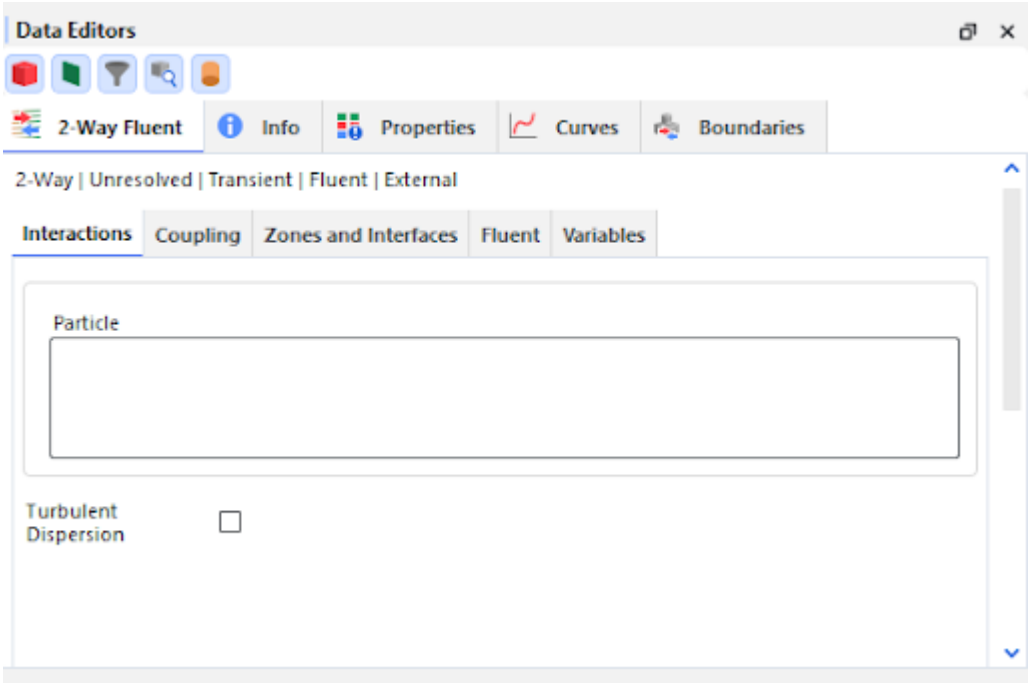
Figure 2.64: CFD Coupling entry in the data tree has 2-way Fluent coupling selected.



7. Open the 2-Way Fluent data panel and edit the coupling setup as usual.

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

Figure 2.65: The Unresolved 2-way coupling mode is enabled, and the coupling setting can usually be edited.

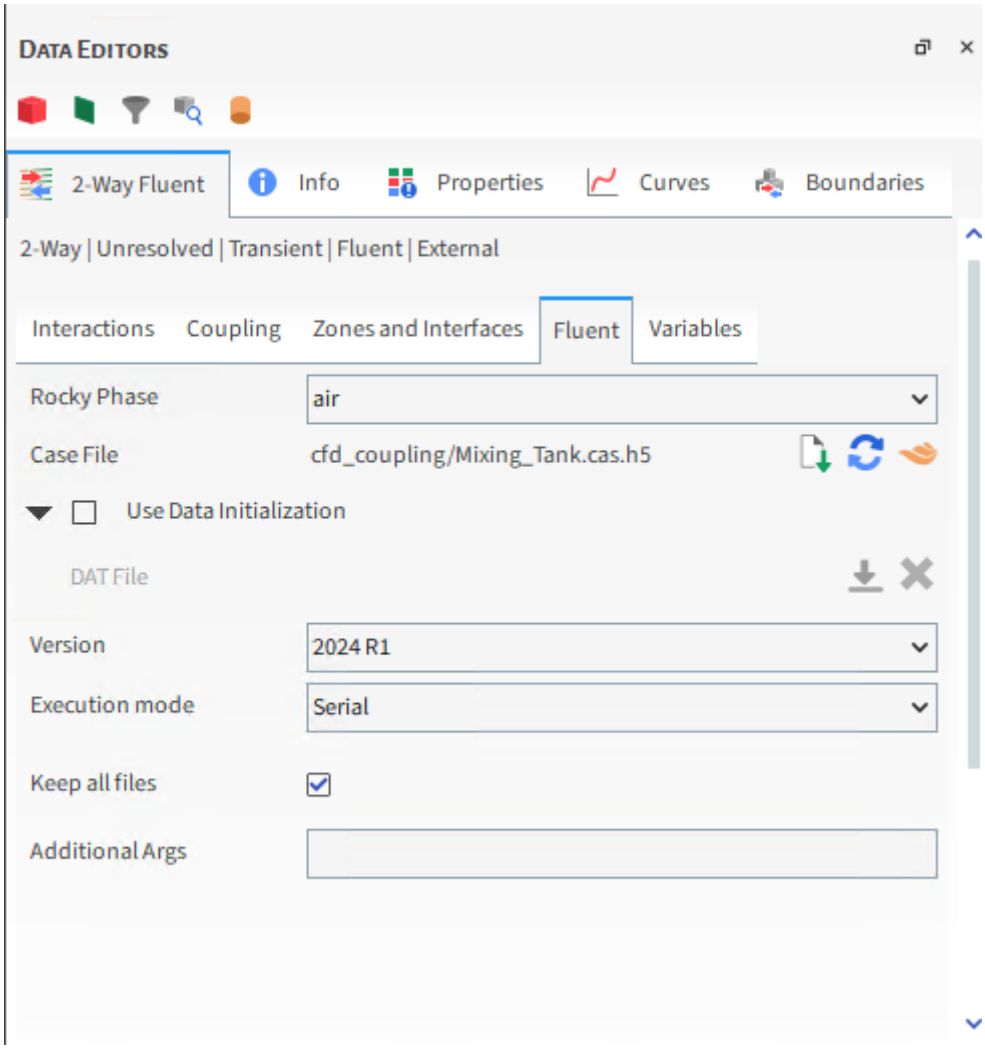


8. Set up the Fluent execution setting - select to run either in serial or parallel and the number of CPU processes that are used to solve CFD equations.

   Important: As the Fluent run is controlled by Rocky in the 2-way coupling, the execution settings defined in the Rocky setup are used.

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

Figure 2.66: The Fluent tab allows the definition of the execution mode and the data used as the initial fluid field.



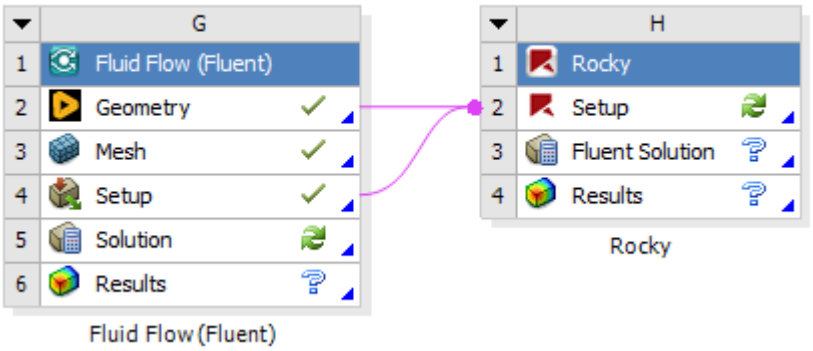
9. For prescribing an initial fluid field for the CFD solution, the .dat file should be imported at the Rocky setup.

   Important: The current version does not allow the initial flow field to be imported from another Fluent run within the workbench. The fluent data file must be imported using the Fluent tab in Rocky.

10. For sharing the geometry between projects, link the **Geometry** of the Fluid project to the **Setup** of the Rocky project. Doing so, the geometry will be automatically imported into Rocky.

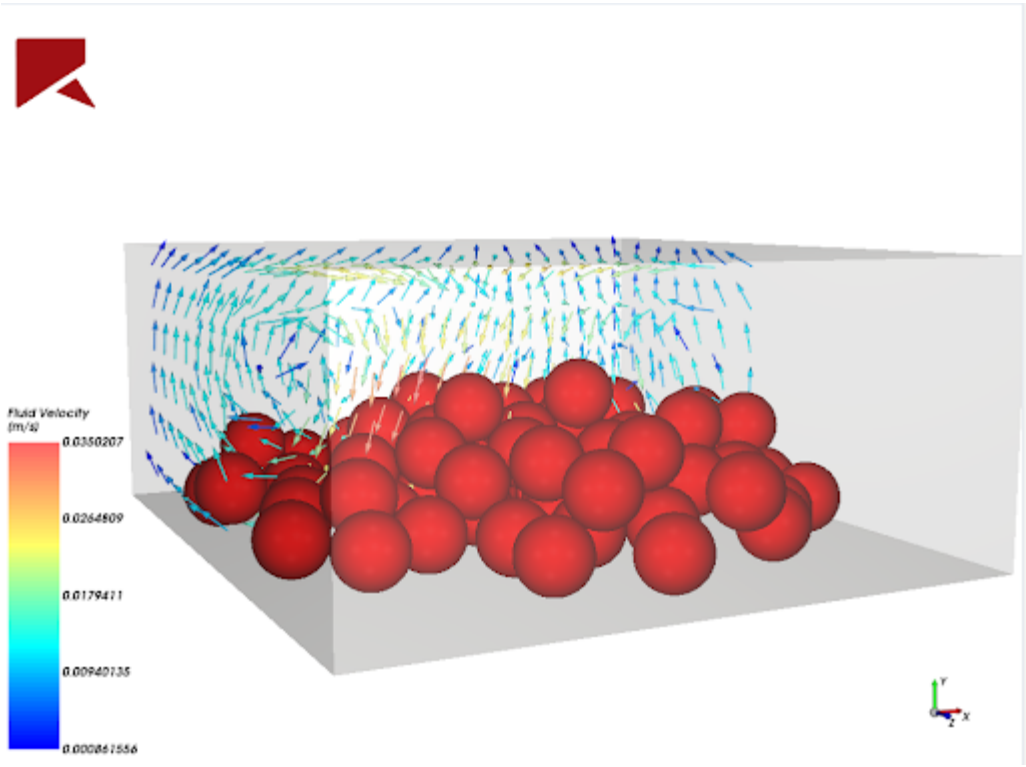
Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

Figure 2.67: Unresolved 2-way coupled simulation with geometry shared between Rocky and Fluent.



11. The CFD and DEM results of the unresolved 2-way coupling can be simultaneously post-processed in Rocky by opening the **Setup** entry of the Rocky project. Fluid quantities such as pressure and velocities can be analyzed and shown in the 3D view together with the particles.

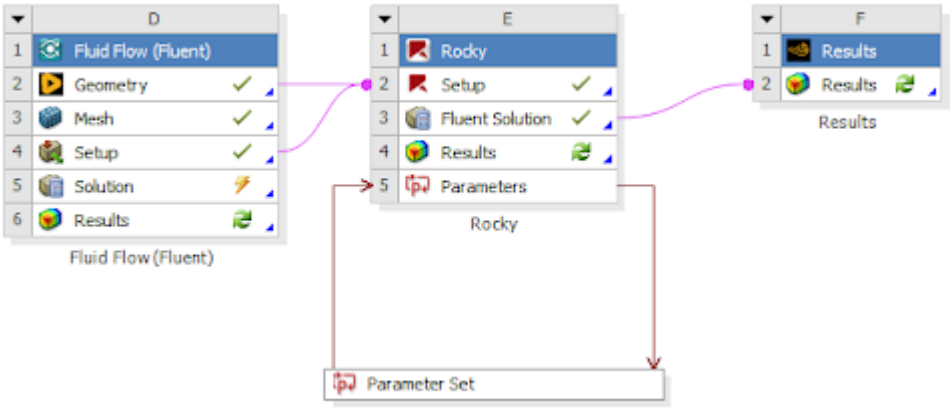
Figure 2.68: Integrated fluid and particles post-processing inside Rocky.



12. The post-processing of additional fluid data can be done inside Fluent. For this purpose, a new **Results** component should be linked to the **Fluent Solution** of the coupled Rocky project.

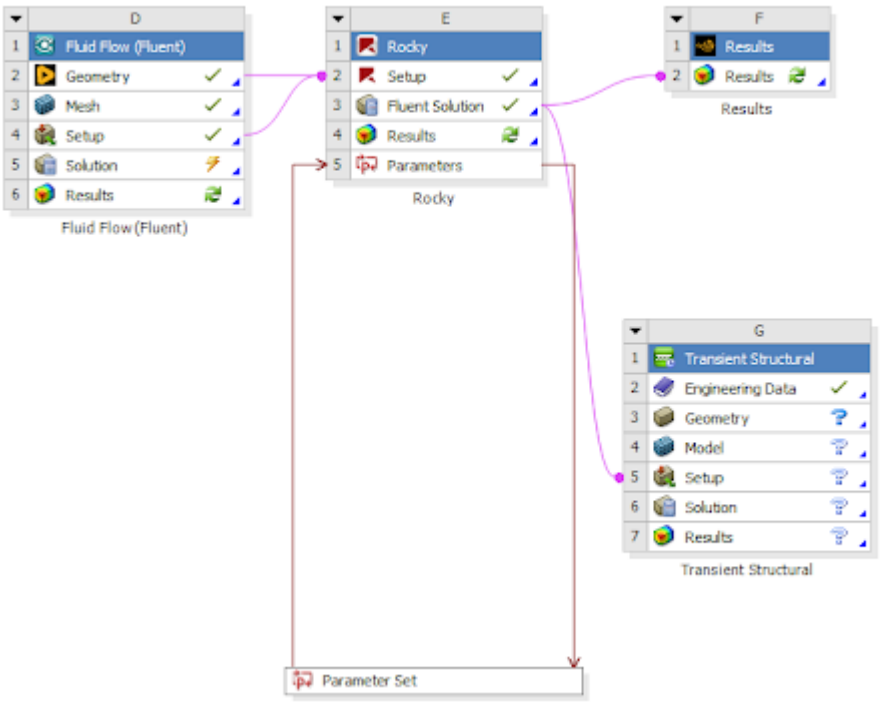
Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

Figure 2.69: The Fluent Solution entry should be used to send the fluid data for external post-processing.



13. The **Fluent Solution** entry in the Rocky project should be used for sending the fluid data to another software, such as a post-processing software or a structural analysis.

Figure 2.70: Input and output parameters created in Rocky are available in Workbench.

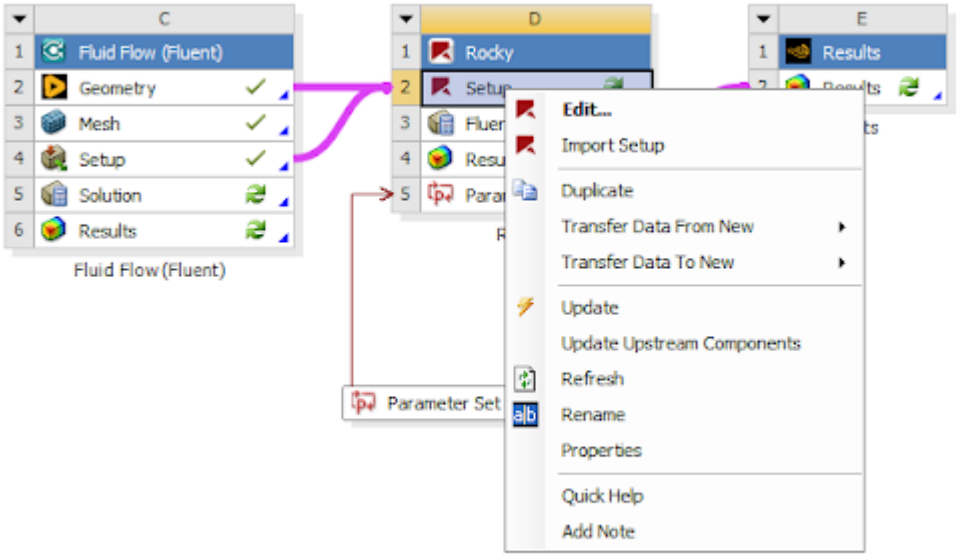


14. For updating changes made to an upstream step, such as changes in a Fluent boundary condition or CFD mesh, select **Refresh** from the Rocky Setup entry.

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

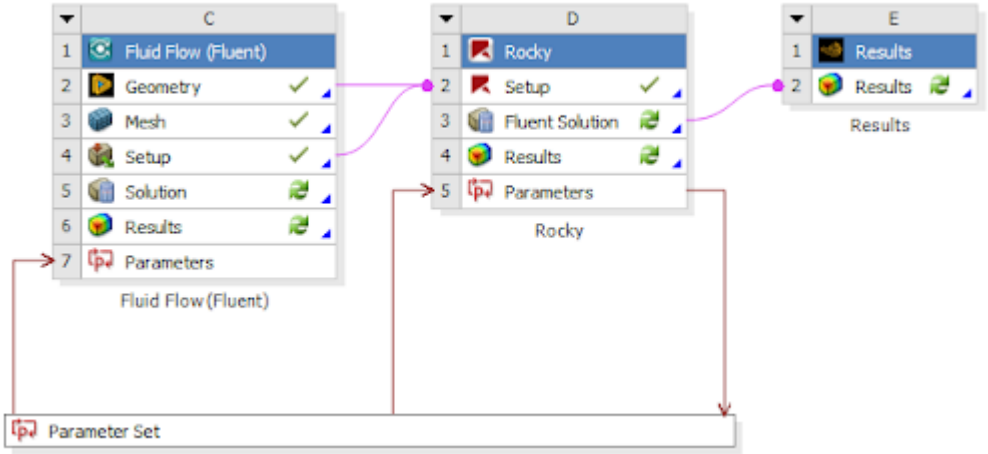
Important: Any change in the Fluent Setup should be made in the Fluent Project that is linked to the Rocky simulation, not from the 2-way Fluent Data Editor in the Rocky UI. The reason is that the CFD setup from the Fluid Flow project will be copied and used as the base setup for the coupled simulation. Any changes made using Rocky will be overwritten when updating the workbench project.

Figure 2.71: If something is changed in an upstream component, the Refresh option in the Setup entry should be used to update the Fluent project used in the coupled simulation.



15. Inputs and Outputs parameters of the DEM simulation are exposed in the workbench as any other Ansys solver and can be used to evaluate several design points automatically.

Figure 2.72: The Fluent solution entry should be used for sharing the fluid data with another software, such as for structural analysis.

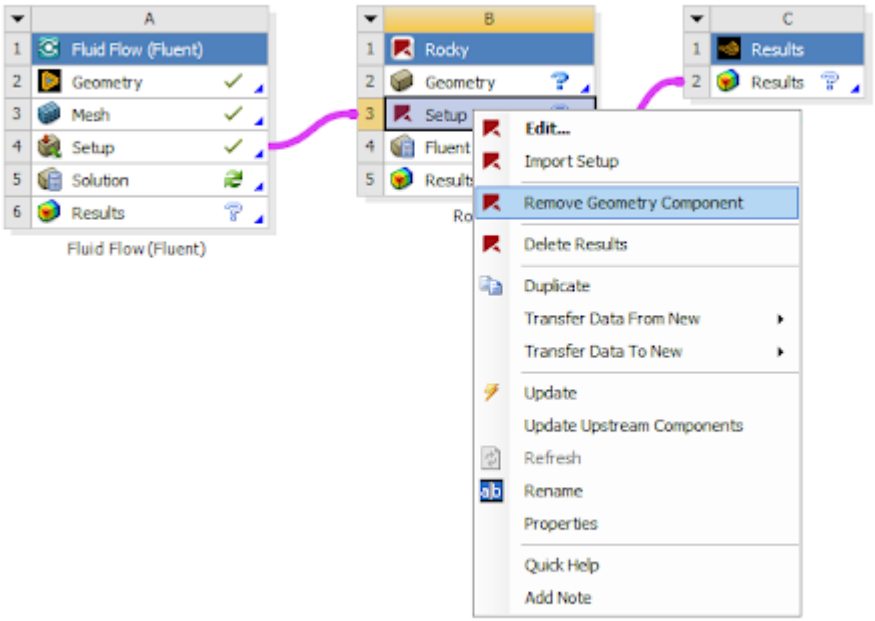


Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

SEMI-RESOLVED 2-WAY ANSYS FLUENT COUPLING WITH WORKBENCH

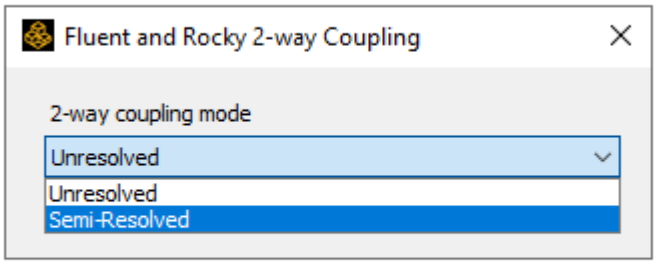
1. Drag the Fluent analysis and set up the Fluent Project.
2. Drag a Rocky Analysis to the Project Schematics. Do not drop it on top of the Fluent setup.
3. If the DEM simulation has no walls, you can Remove the Geometry Component.

Figure 2.73: Remove Geometry component.



4. Select Setup in the Rocky Project and click Edit. Select **Semi-Resolved** as the 2-way coupling mode.

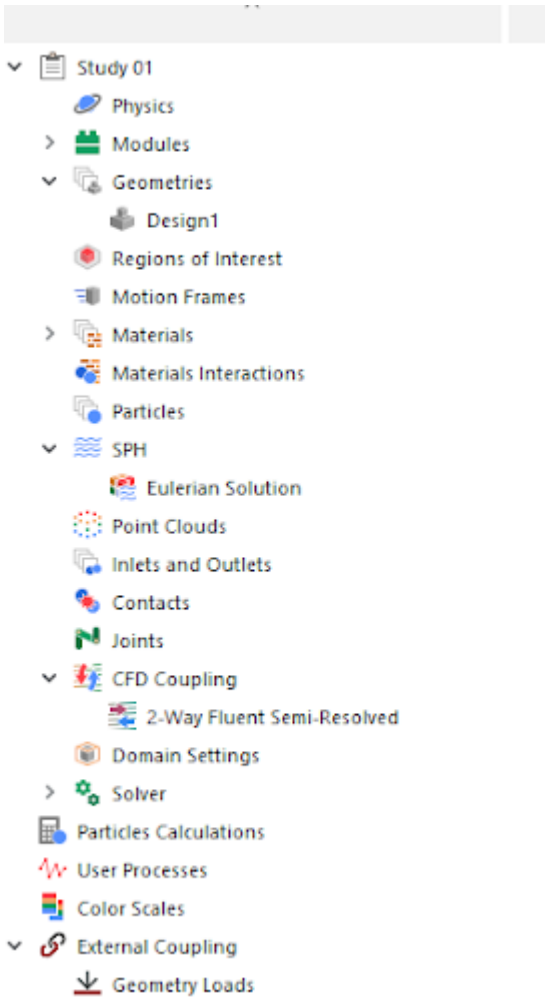
Figure 2.74: Semi-Resolved mode selected.



Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

5. When opening Rocky, the CFD Coupling entry at the Data Tree has the 2-Way Fluent Semi-Resolved mode selected.

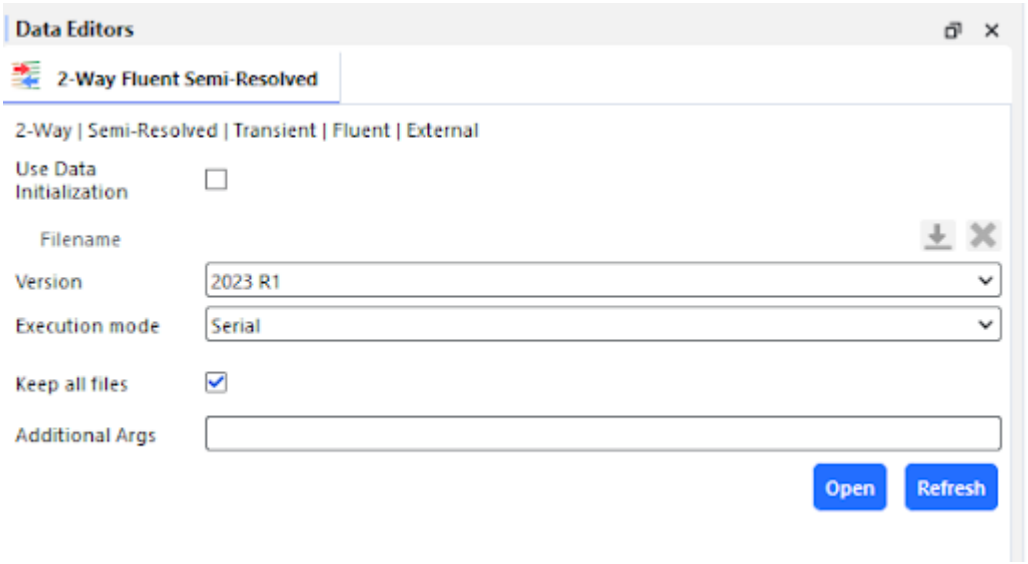
Figure 2.75: 2-Way Fluent Semi-Resolved mode selected.



6. In the **2-Way Fluent Semi-Resolved Data Editor**, the coupling setup can be edited as usual.

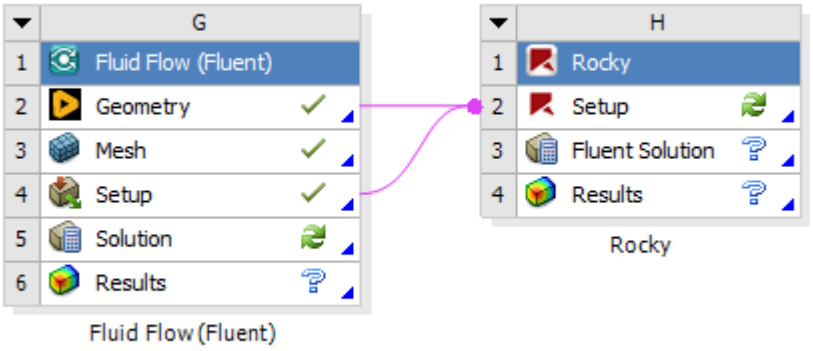
Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

Figure 2.76: 2-Way Fluent Semi-Resolved settings.



7. For sharing the geometry between projects, link the geometry between the Fluid project and the Rocky Setup projects. Doing so, the geometry will be automatically imported into Rocky.

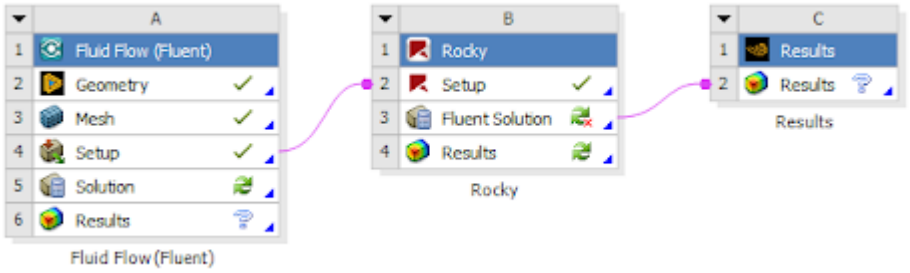
Figure 2.77: Sharing the geometry between projects.



8. For the Semi-Resolved coupling, fluid data cannot be post-processed in Rocky. For post-processing fluid data, add a new **Results** component system and link it to the Rocky project **Fluent Solution**.

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

Figure 2.78: Post-processing fluid data in 2-way semi-resolved coupled runs.



2.3.2. Ansys Rocky and Ansys Mechanical

Rocky's integration with Ansys Mechanical enables Mechanical to conduct either transient or static structural analyses based upon the particle forces that Rocky calculates. These two tools can operate as connected components within Ansys Workbench, or the DEM portion can be done separately in Rocky and the results later imported into Workbench for the structural analysis portion.

This section covers the following topics:

1-WAY COUPLING WITH MECHANICAL (p. 170)

2-WAY THERMAL COUPLING WITH MECHANICAL (p. 172)

2-WAY STRUCTURAL COUPLING WITH MECHANICAL (p. 189)

1-WAY COUPLING WITH MECHANICAL

Ansys Products needed: Ansys Rocky and Ansys Workbench

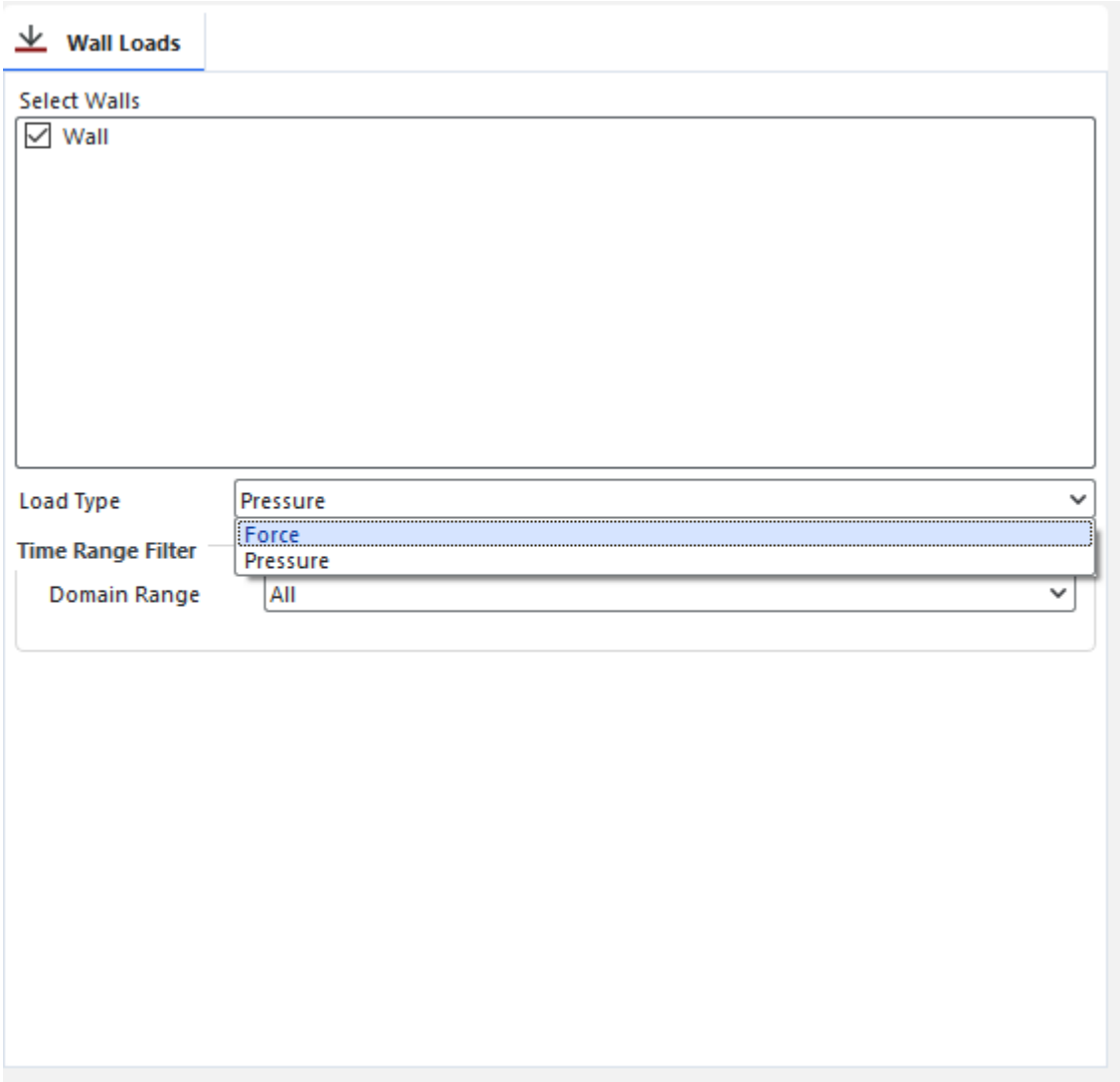
For projects coupled specifically with Ansys Mechanical, opening the **Wall Loads** in the **Data Editors** panel enables you to select which Rocky data items you want transferred to the Workbench connected Mechanical program.

For example, you can select which Walls you want to analyze in Mechanical, i.e, which loads are to be transferred and which load type you want between Force and Pressure.

See the image below:

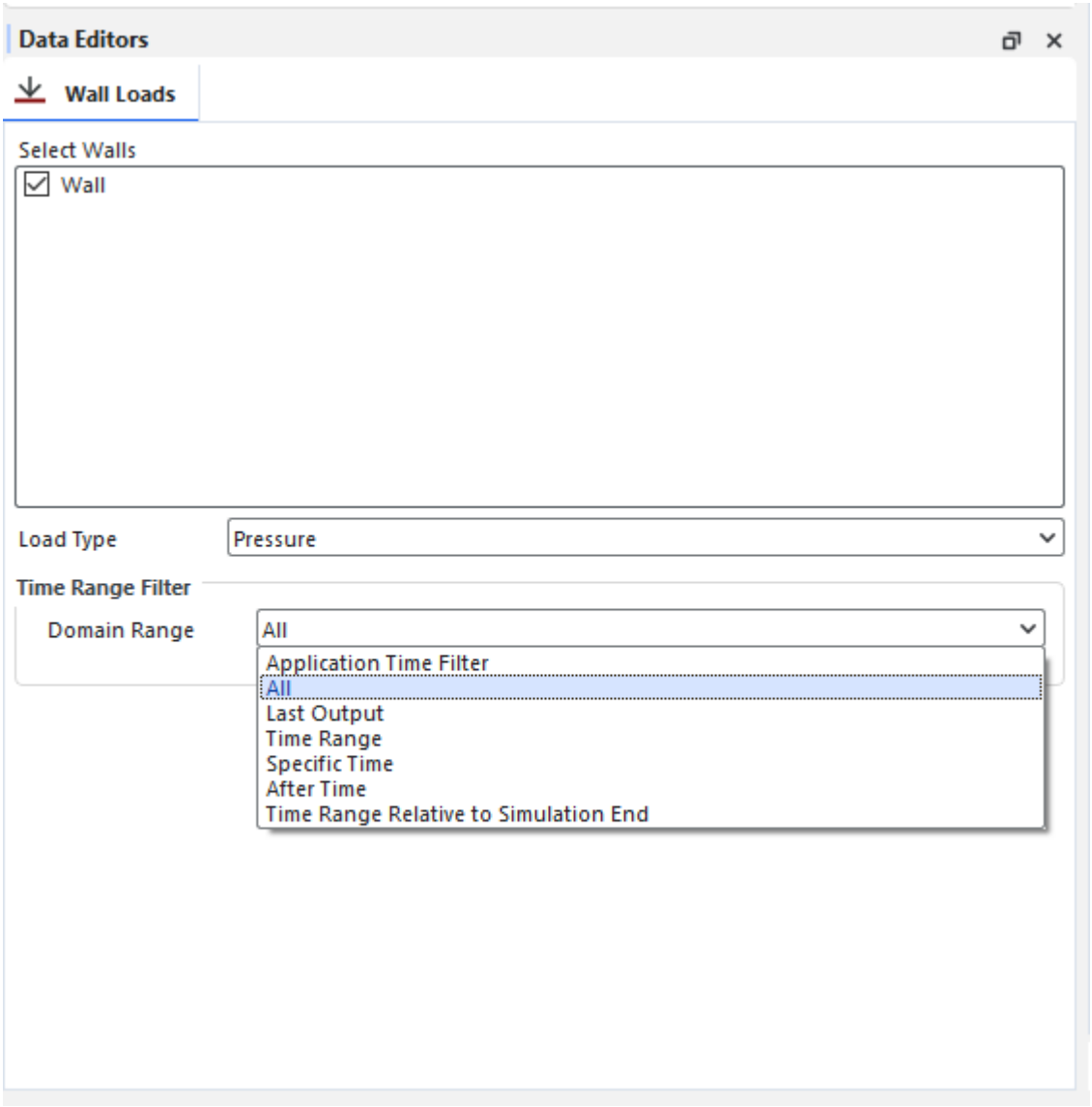
Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

Figure 2.79: Export options for 1-way coupling with mechanical



You can then select the results domain of the analysis that will be exported by setting the time range.

Figure 2.80: Time Range Filte



Important:

When making a 1-way coupling with Mechanical (both 1-Way HTC and 1-Way Static and Transient Structural) the connection between Rocky Geometry and Results to Mechanical Geometry and Setup must be done manually, instead of dragging the Mechanical system and dropping it on top of the Rocky one. Otherwise, the coupling may not occur properly. This process is needed because now, for both 1-Way Static and Transient Structural (Forces only) and 1-Way HTC coupling between Rocky and Mechanical, the data from Rocky are automatically sent to Mechanical, and the manual process to copy and paste the .csv results inside Mechanical is no longer needed. For Pressure in the 1-Way Static and Transient Structural, the copy and paste is still needed.

2-WAY THERMAL COUPLING WITH MECHANICAL

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

[HOW-TO VIDEO]Ansys Rocky: 2-Way Thermal Coupling with Mechanical: https://www.youtube.com/watch?v=DKuTeF6eDG4&list=PL0lZXwHtV6Omiv62KRuPbnZ4oBK8BmDit&index=10

Ansys Products needed: Ansys Rocky, Ansys Mechanical, Ansys Workbench, System Coupling and Ensight.

With this coupling is possible to send heat from Rocky and receive back the temperature from Mechanical using the System Coupling.

To do this, the users are able to create a coupled wall inside Rocky and the System Coupling will identify this walls as regions where the coupling is going to happen.

Limitations:

The coupled walls cannot have:

- Motion Frame
- Translation
- Rotation
- Mass tab
- Wear tab
- Replication tab

1- SETUP IN ANSYS ROCKY

1. Set up your simulation as you wish, then you will have to stipulate within Rocky which geometry of your simulation you want to couple.

   To do this, right-click on the desired geometry, then click on **Convert to** and then on **System Coupling Wall**.

   Figure 2.81: Convert Geometry to System Coupling Wall

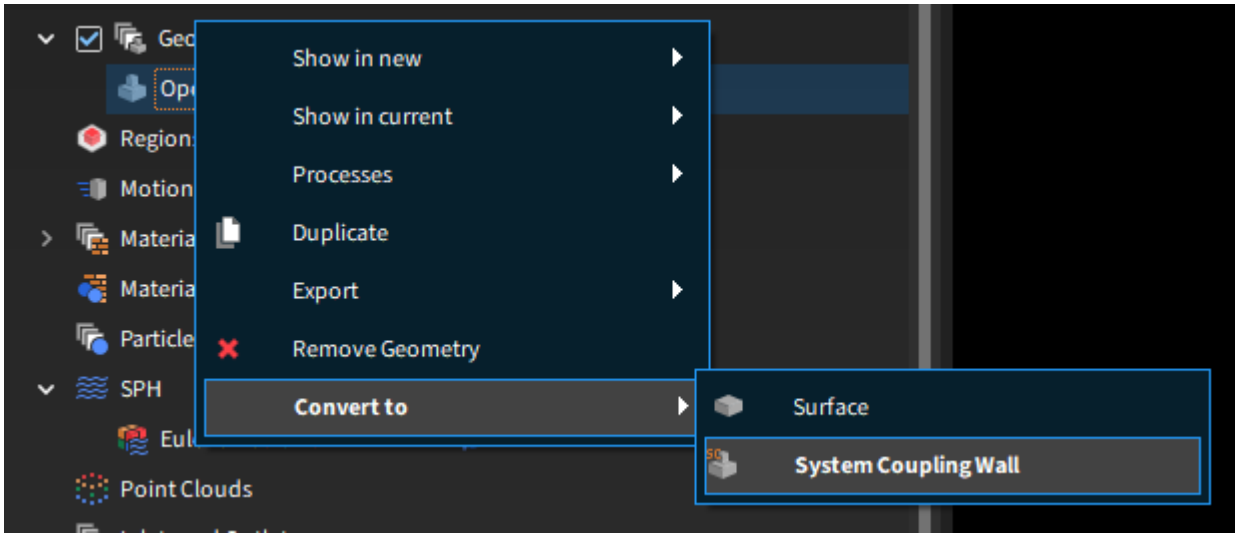
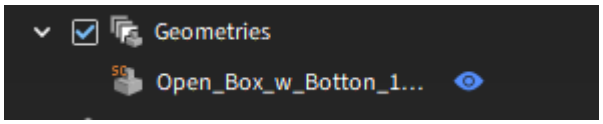


Figure 2.82: System Coupling Wall Data Panel



2. Click on **File** and then on **Write System Coupling file**. This step creates a file with information about where your Rocky file is for System Coupling.

Figure 2.83: Write System Coupling File Rocky

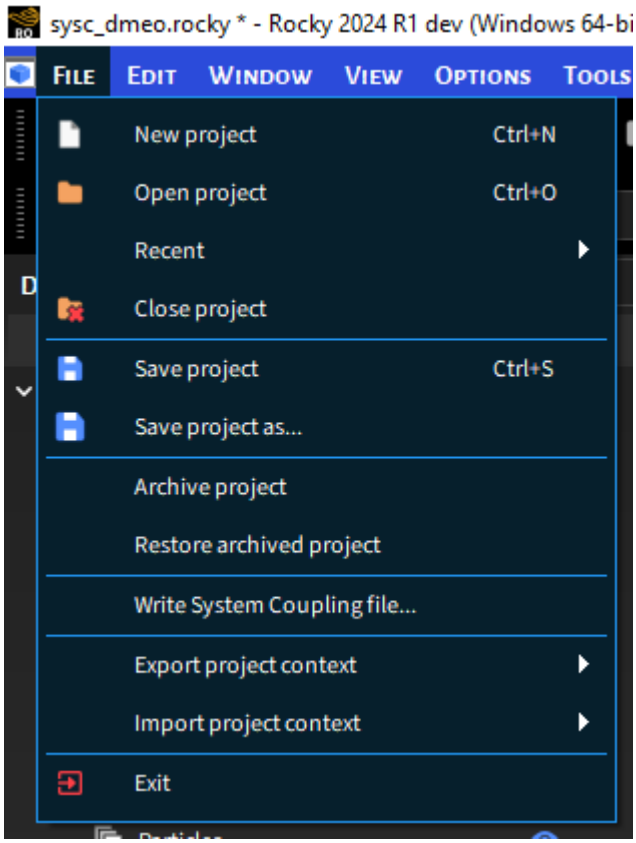


Figure 2.84: System Coupling Participant file

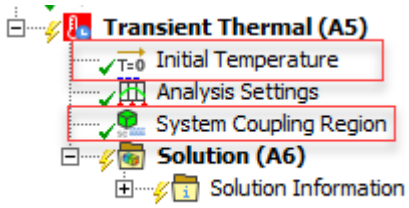


2- SETUP IN ANSYS MECHANICAL

1. Open your project in Ansys Mechanical and set the **Initial Temperature** and the **System Coupling Region** using the **Transient Thermal** option.

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

Figure 2.85: Transient Thermal Ansys Mechanical



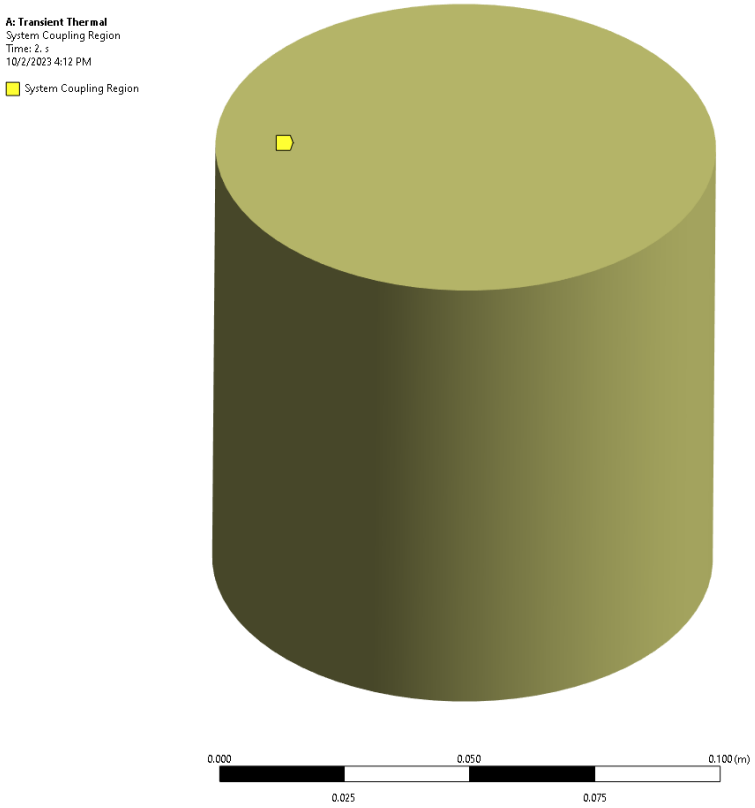
2. Set the System Coupling Region, as shown in the figures below:

Figure 2.86: Details of System Coupling Region

Details of "System Coupling Region"

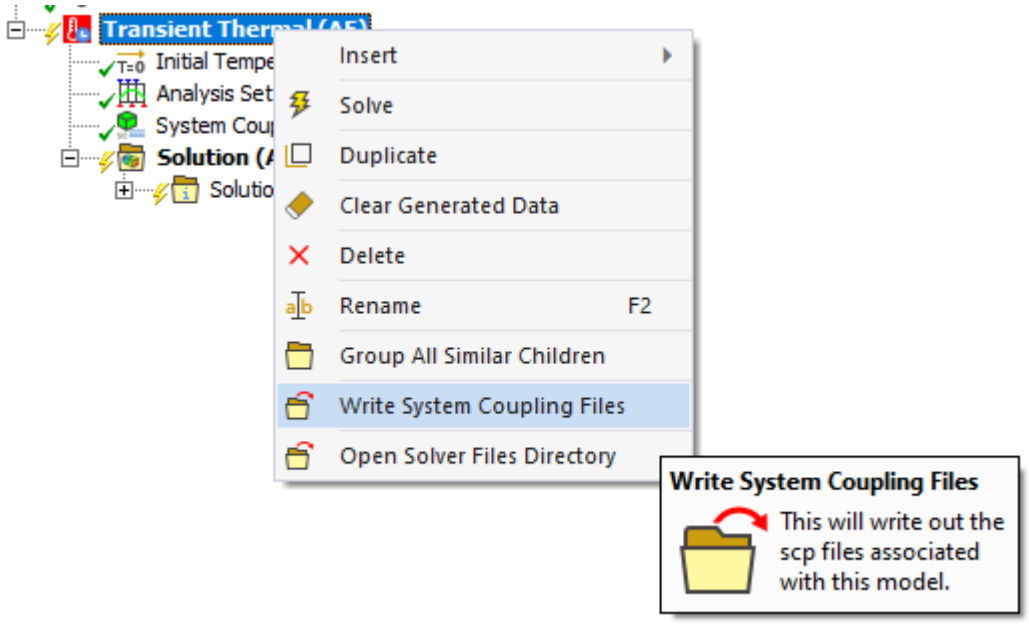
Scope	
Scoping Method	Geometry Selection
Geometry	3 Faces
Definition	
Type	System Coupling Region
Interface Number	1.
Export Results	Yes
Suppressed	No

Figure 2.87: System Coupling Region in Mechanical



3. Export the System Coupling File, by right-clicking on **Transient Thermal** and then on **Write System Coupling Files**.

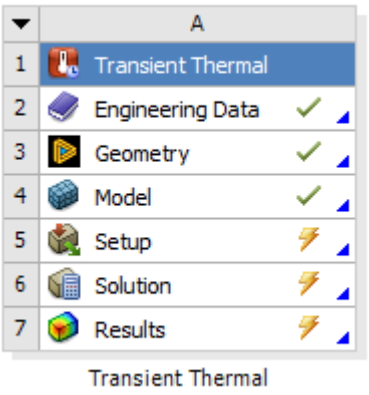
Figure 2.88: Write System Coupling File Mechanical



3- SETUP IN ANSYS WORKBENCH (TRANSIENT THERMAL)

1. Open your project in the Ansys Workbench and set the 2-way Thermal Coupling using the **Transient Thermal** option.

Figure 2.89: Transient Thermal in Workbench



Learn more at About Rocky and Ansys Workbench Integration (p. 144)

4- SETUP SYSTEM COUPLING

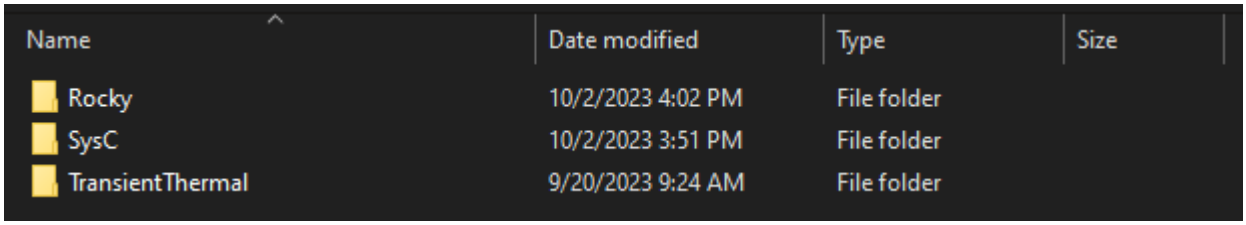
Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

1. Open System Coupling and select a folder of your choice.

Tip:

When you are doing this coupling, it is advisable to create three folders for your project, one for **Rocky**, another for **Workbench** and another for **System Coupling**.

Figure 2.90: Transient Thermal in Workbench

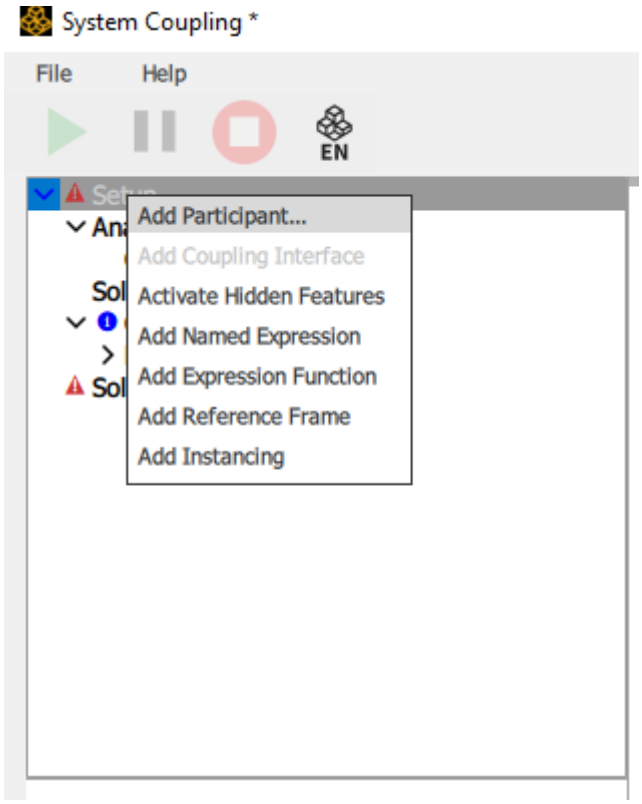


Note:

When add the folder some Warnings will appear in the message tag, but they will be resolved as you do the setup.

2. In the Data Panel, right-click **Setup** and then click **Add Participant**.

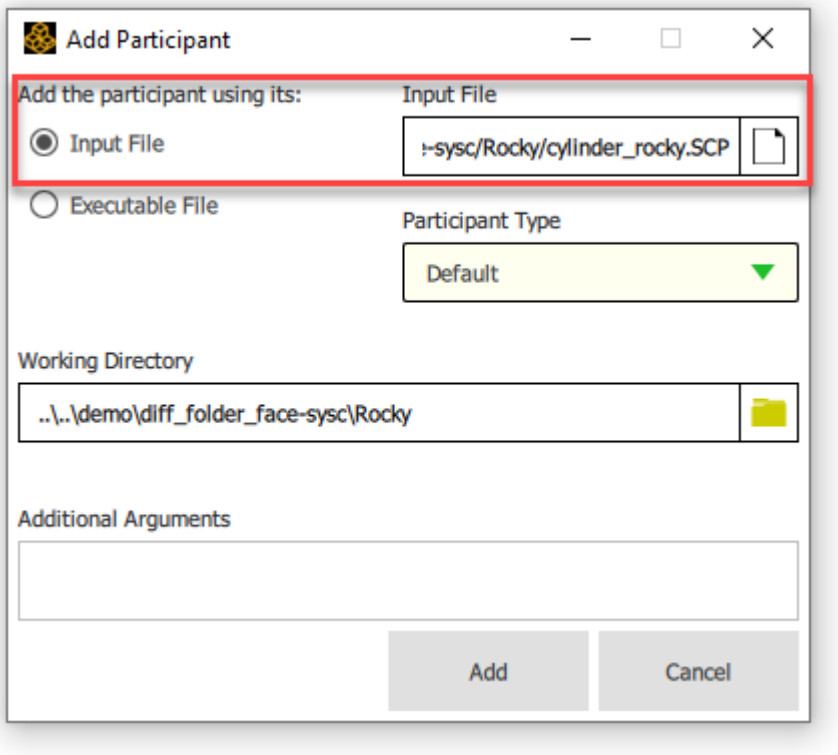
Figure 2.91: Add Participant in System Coupling



Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

3. In the Add Participant window, select **Input File** and add the **System Coupling File** (.SCP) you saved from Rocky.

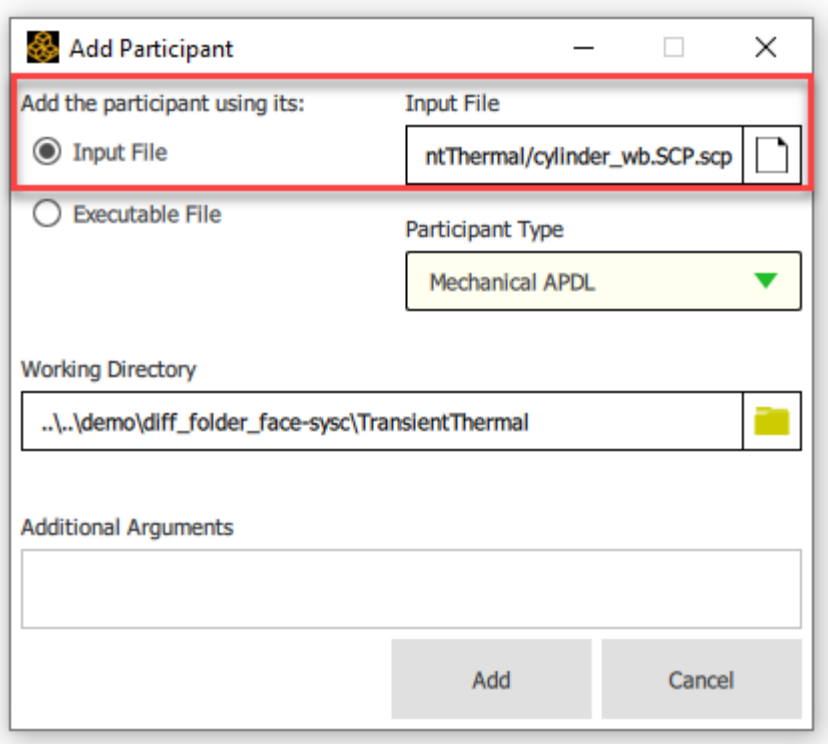
Figure 2.92: Add Participant: System Coupling File Rocky



4. Repeat step 1, and in the Add Participant window, select **Input File** and add the **System Coupling File** (.SCP) you saved from Mechanical.

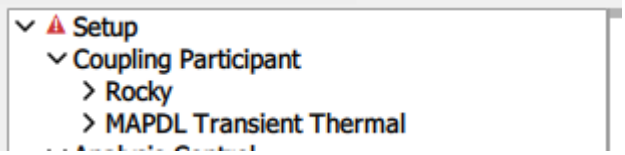
Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

Figure 2.93: Add Participant: System Coupling File Mechanical



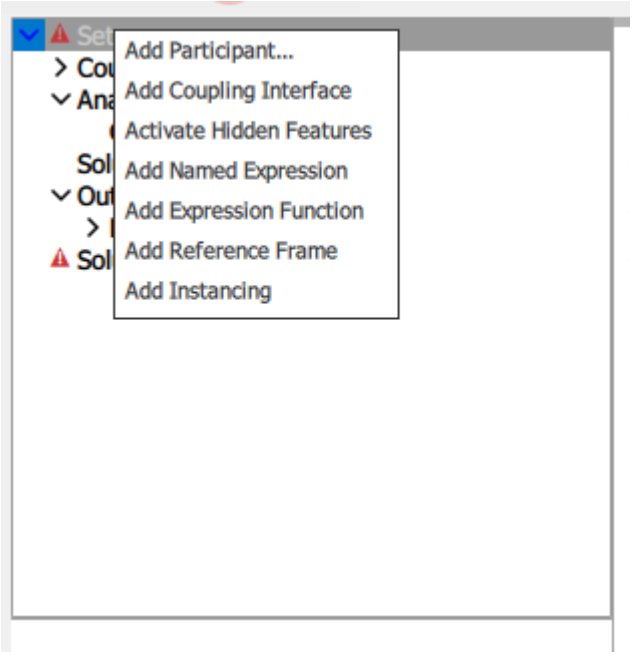
Your Data Panel must have both Rocky and Mechanical files:

Figure 2.94: System Coupling Participants



5. In the Data Panel, right-click **Setup** and then click **Add Coupling Interface**.

Figure 2.95: Coupling Interface



6. Right Click on **Coupling Interface** and then click on **Add Data Transfer**.

Figure 2.96: Coupling Interface Data Panel

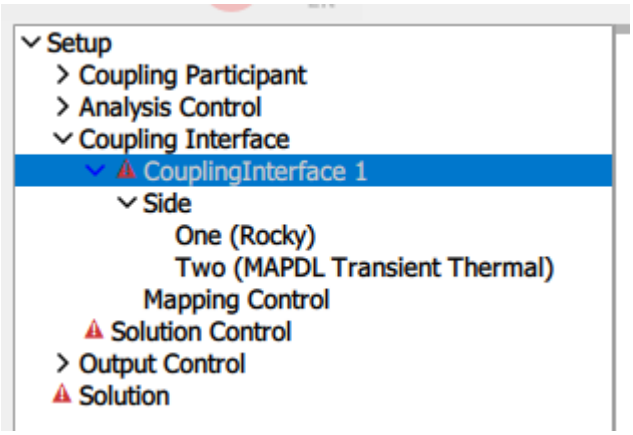
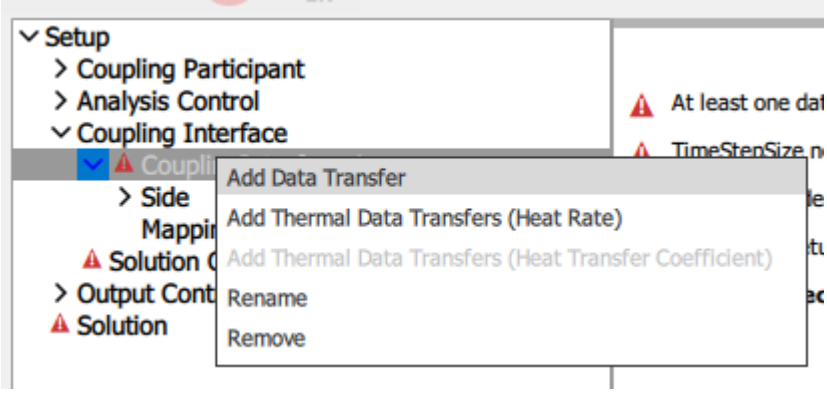


Figure 2.97: Add Data Transfer



Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

7. Set the **Data Transfer** to Rocky as show in the figures below:

Figure 2.98: Data Transfer Rocky

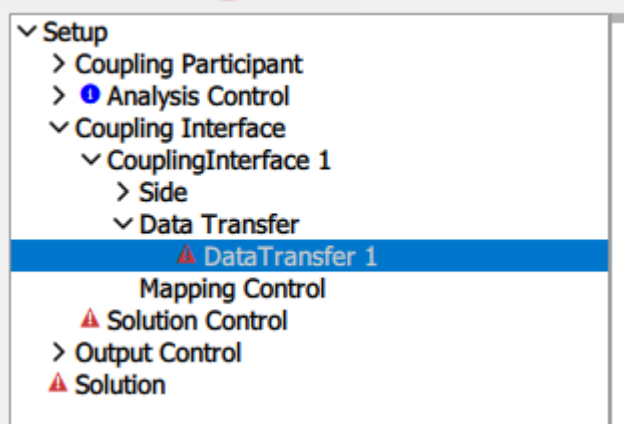
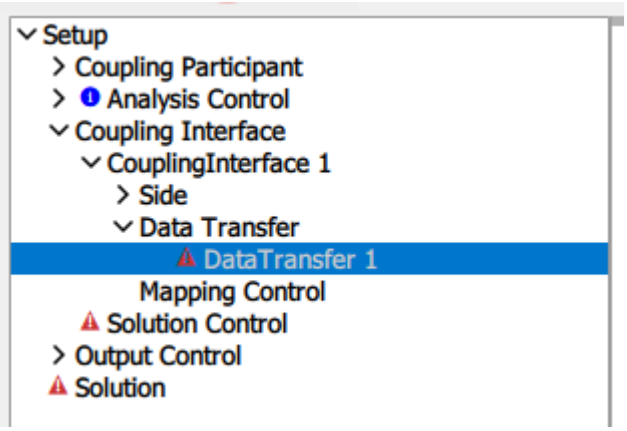


Figure 2.99: Data Transfer Rocky Setup (Temperature)

Internal Name	DataTransfer 1
Suppress	False
Target Side	One (Rocky)
Option	UsingVariable
Source Variable	Heat_Flow
Target Variable	Temperature
Ramping Option	None
Relaxation Factor	1
Convergence Target	0.01
Mapping Type	ProfilePreserving
Unmapped Value Option	Nearest Value

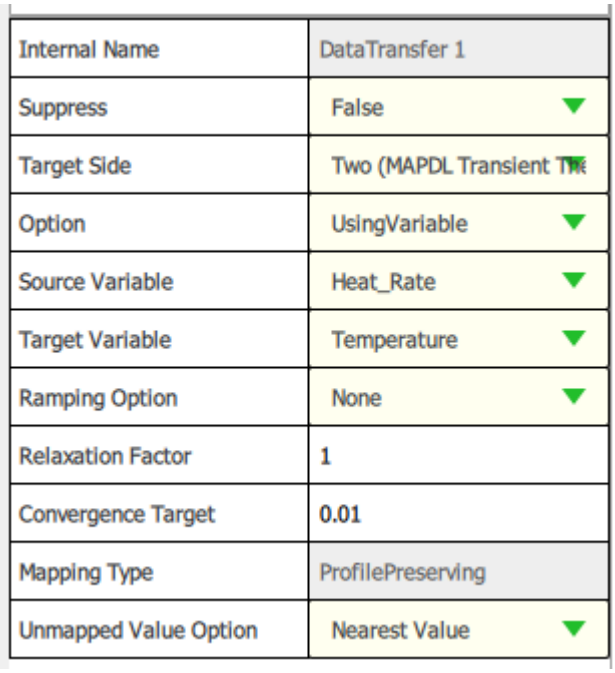
8. Repeat step 6, and set the **Data Transfer** to Mechanical as show in the figures below:

Figure 2.100: Data Transfer Mechanical



Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

Figure 2.101: Data Transfer Mechanical Setup (Heat Flow)

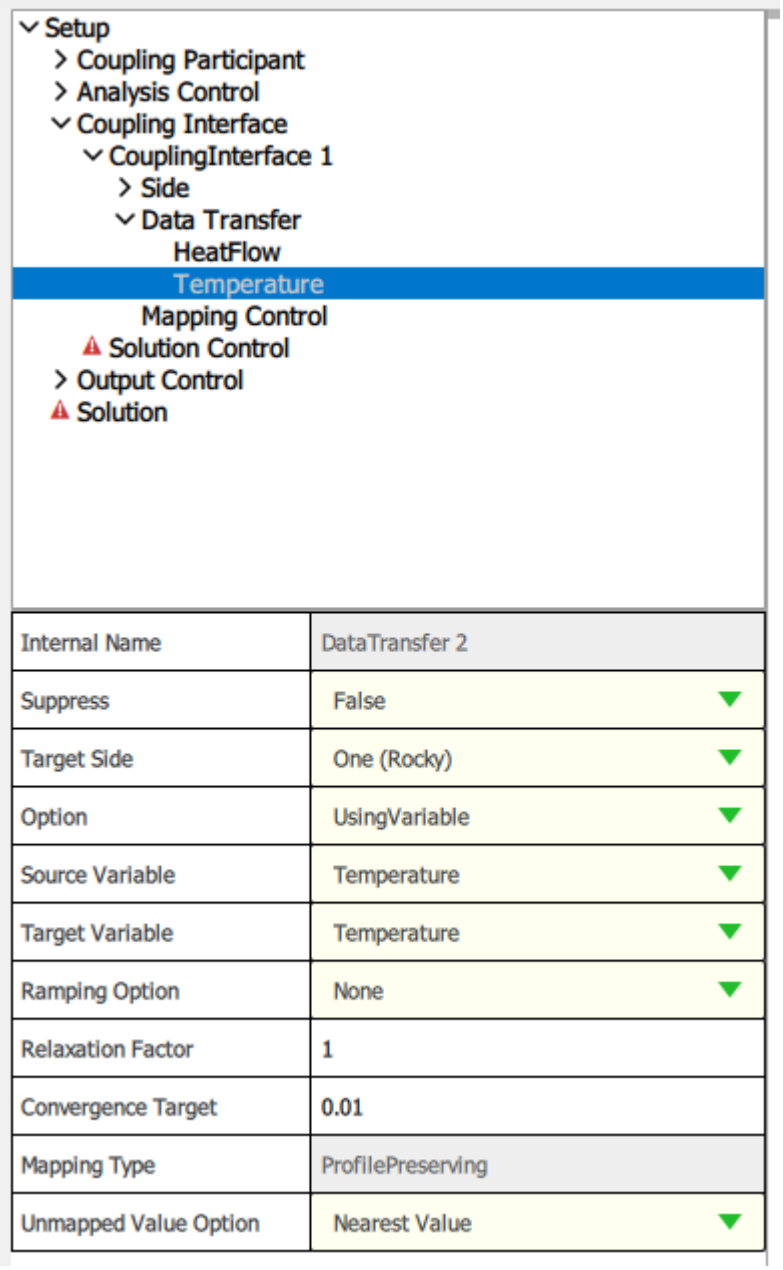


Internal Name	DataTransfer 1
Suppress	False
Target Side	Two (MAPDL Transient Th
Option	UsingVariable
Source Variable	Heat_Rate
Target Variable	Temperature
Ramping Option	None
Relaxation Factor	1
Convergence Target	0.01
Mapping Type	ProfilePreserving
Unmapped Value Option	Nearest Value

In the example below the Rocky Data Transfer is named as Temperature and the Mechanical Data Transfer is named as Heat Flow:

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

Figure 2.102: Data Transfer Rocky and Mechanical



9. In **Solution Control** define the **End Time** and the **Time Step Size**.

Important:

The **End Time** and the **Time Step Size** that you need to set in System Coupling must follow this approach:

- The **End Time** must be the same as the **Time** used in Mechanical, and the **Simulation Duration** in Rocky;

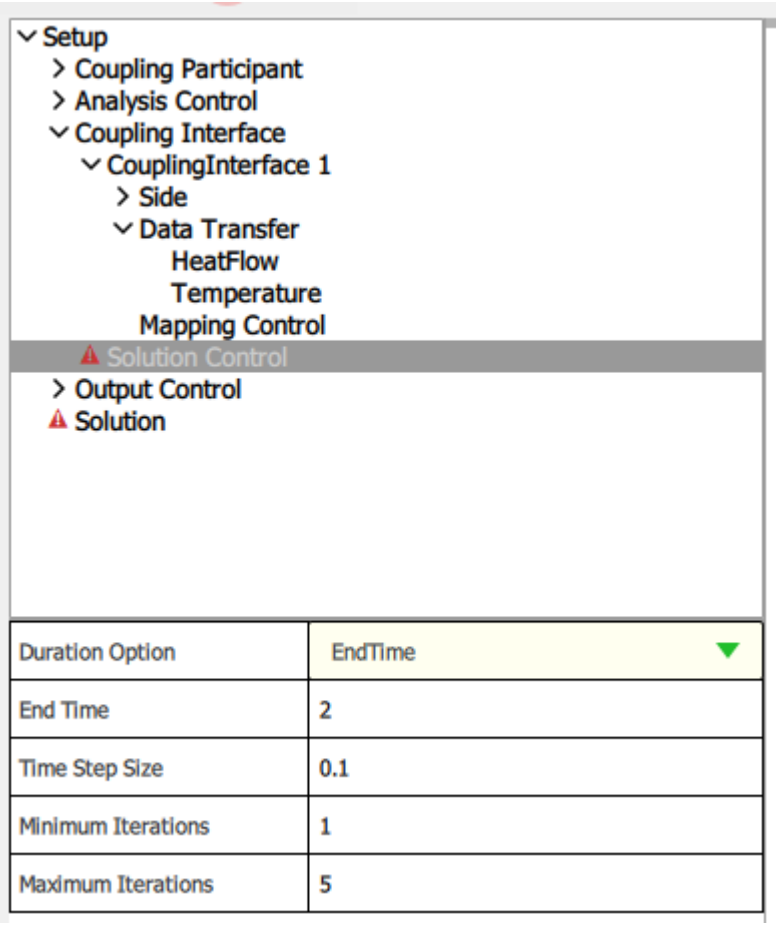
Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

- And **Time Step Size** could be the same as the **Time Step** (used in Mechanical) and the **Time Interval** (used in Rocky).

Note:

From the 2025 R2 forward, this **End Time** is set automatically.

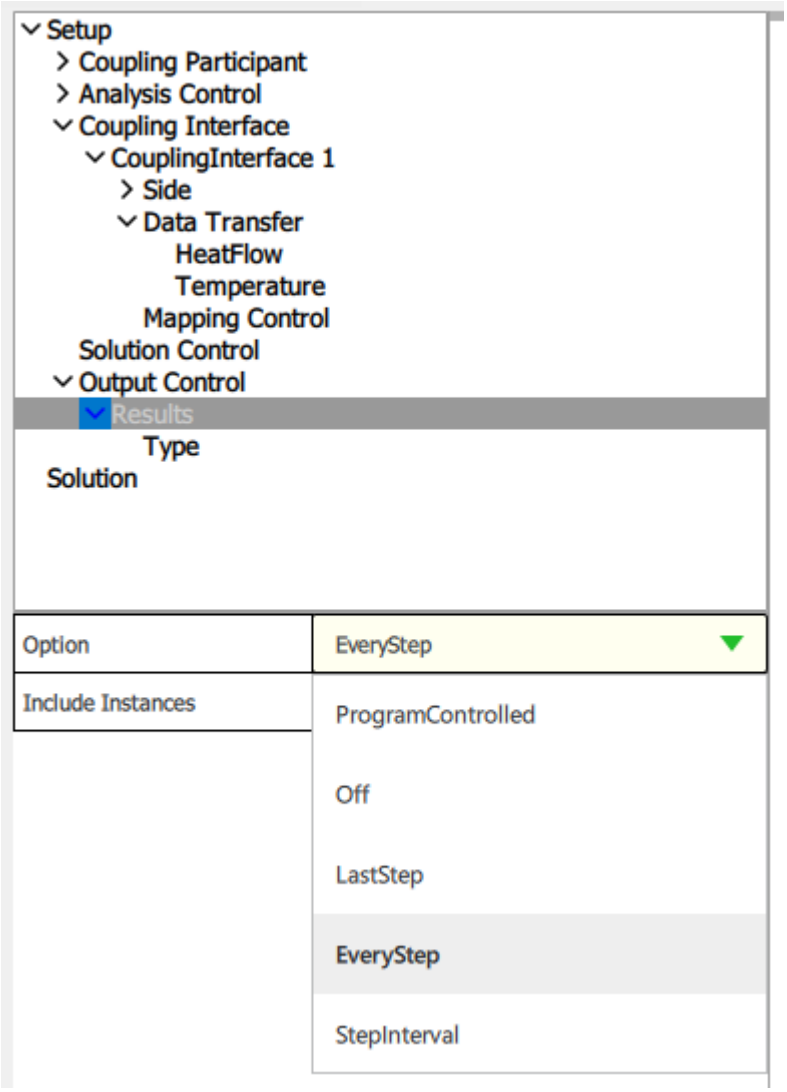
Figure 2.103: Solution Control



10. In **Output Control**, click on **Results**, and then on **Type** and select the **Every Step** option, as shown in the figure below:

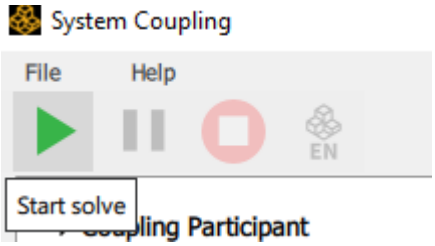
Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

Figure 2.104: Output Control



11. Click on **Start Solve**.

Figure 2.105: Start Solve



5- POST PROCESSING ENSIGHT

1. After solving your simulation, the post processing fase is carried out using the EnSight. Click on **EnSight:**

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

Figure 2.106: System Coupling After Solving

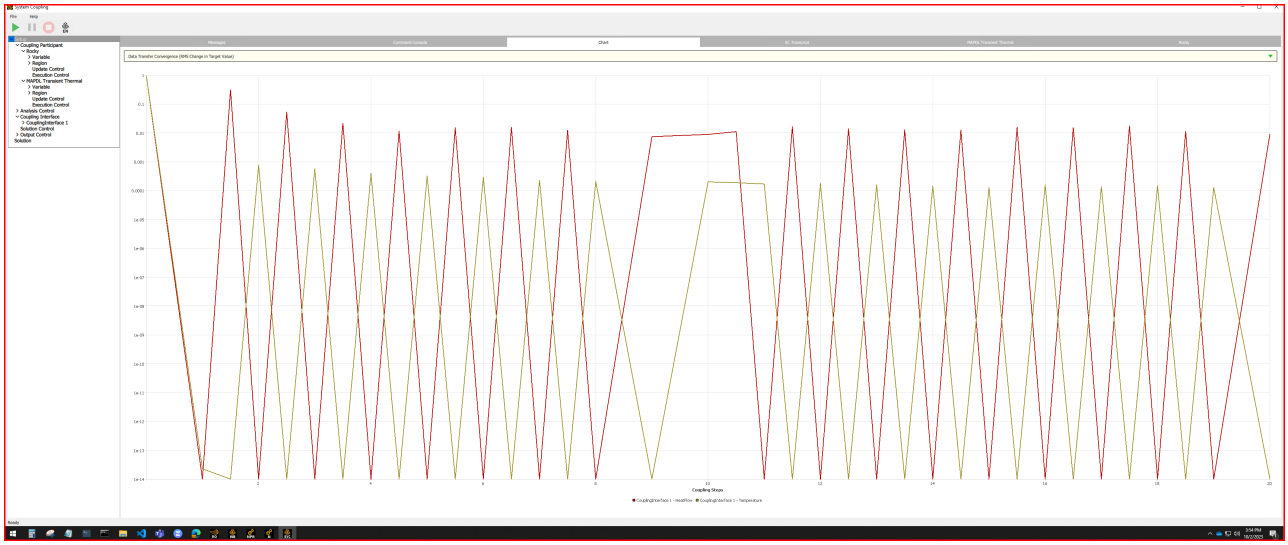
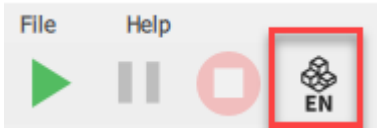
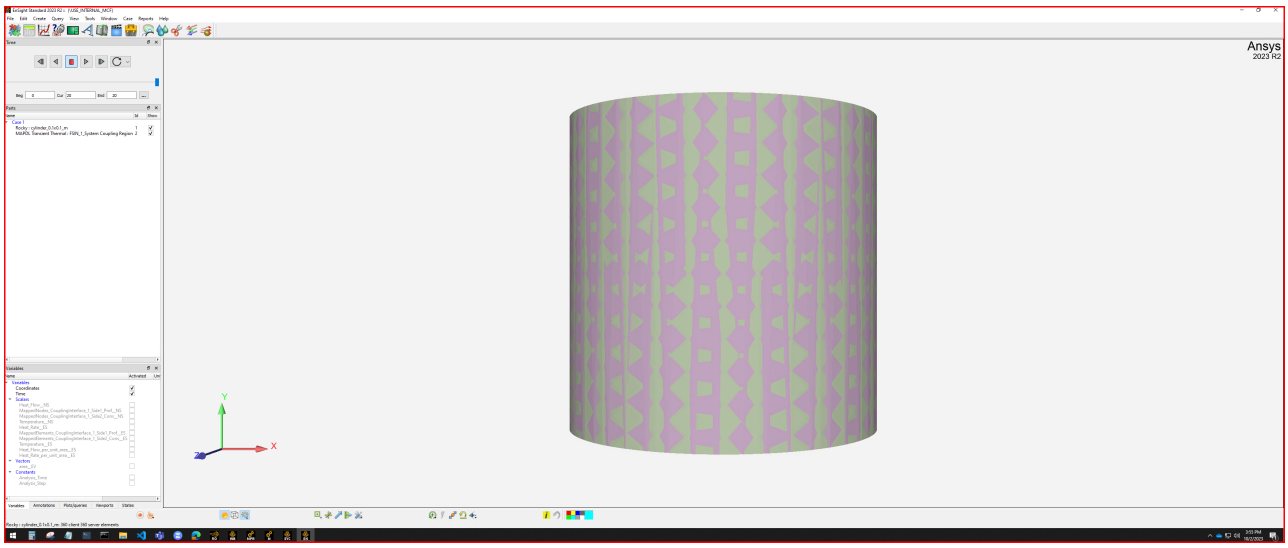


Figure 2.107: Access EnSight through System Coupling



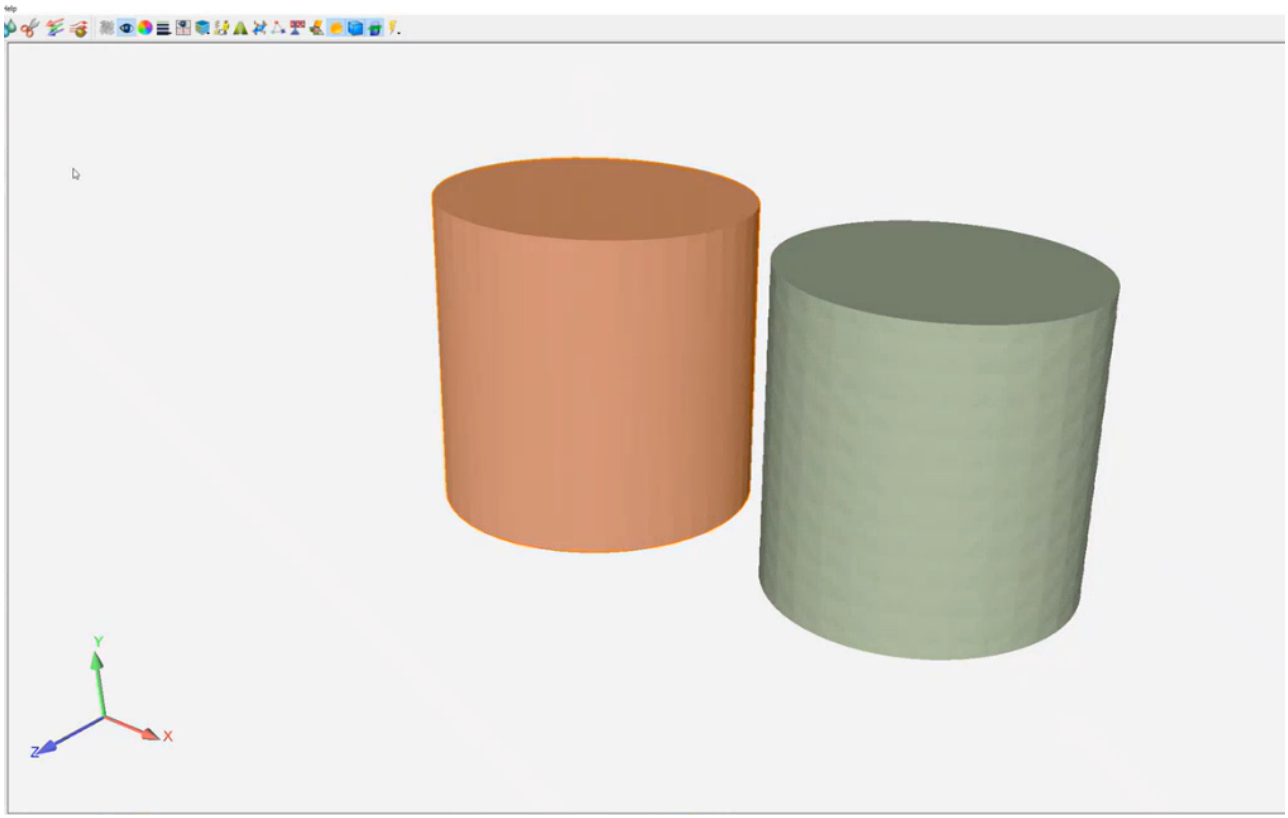
2. The geometries will appear in EnSight as shown in the figure below, one on top of the other, if you prefer you can separate them:

Figure 2.108: Post Processing EnSight



Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

Figure 2.109: Changing the Translation



3. To see the results you have to drag the desired parameter to the geometry you want to see the result:

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

Figure 2.110: Dragging the desired parameter to the Geometry

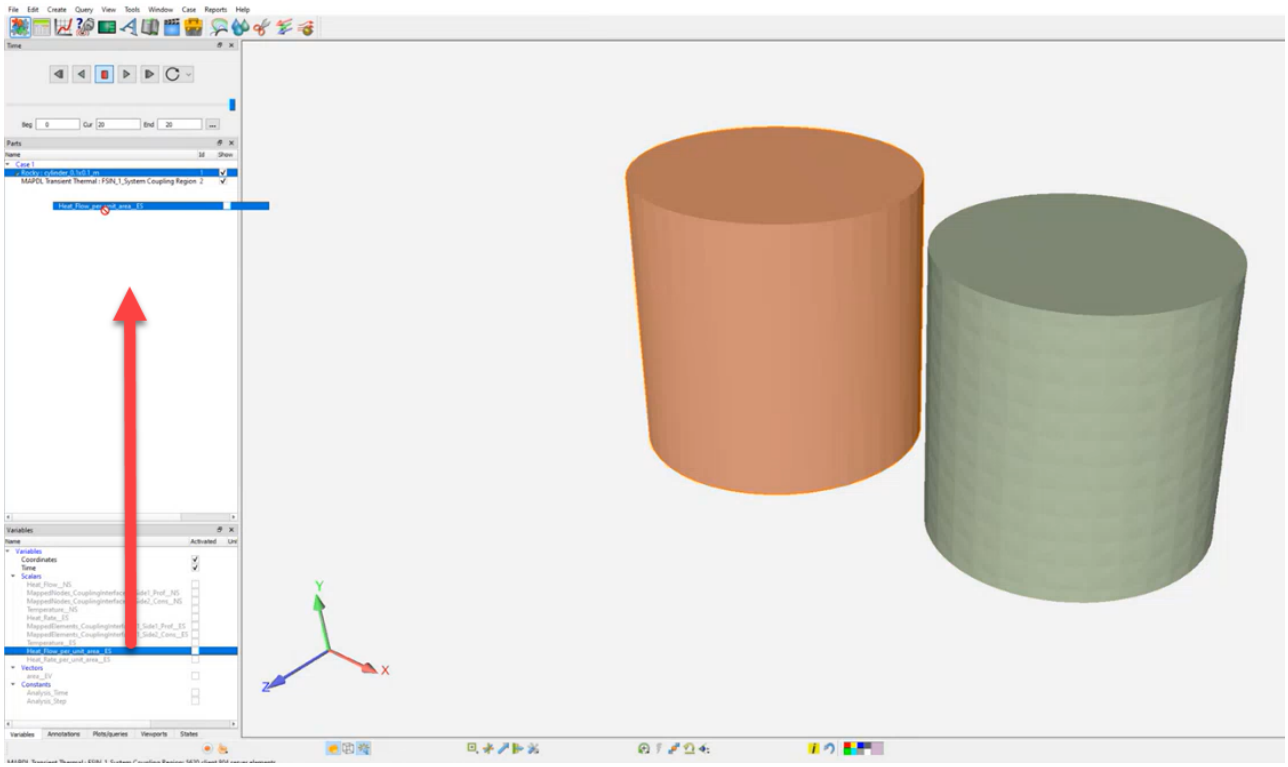
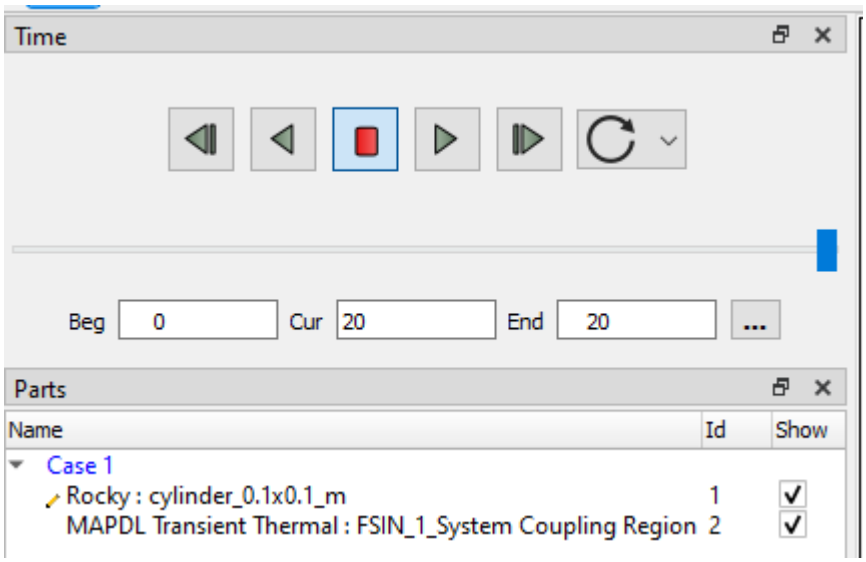
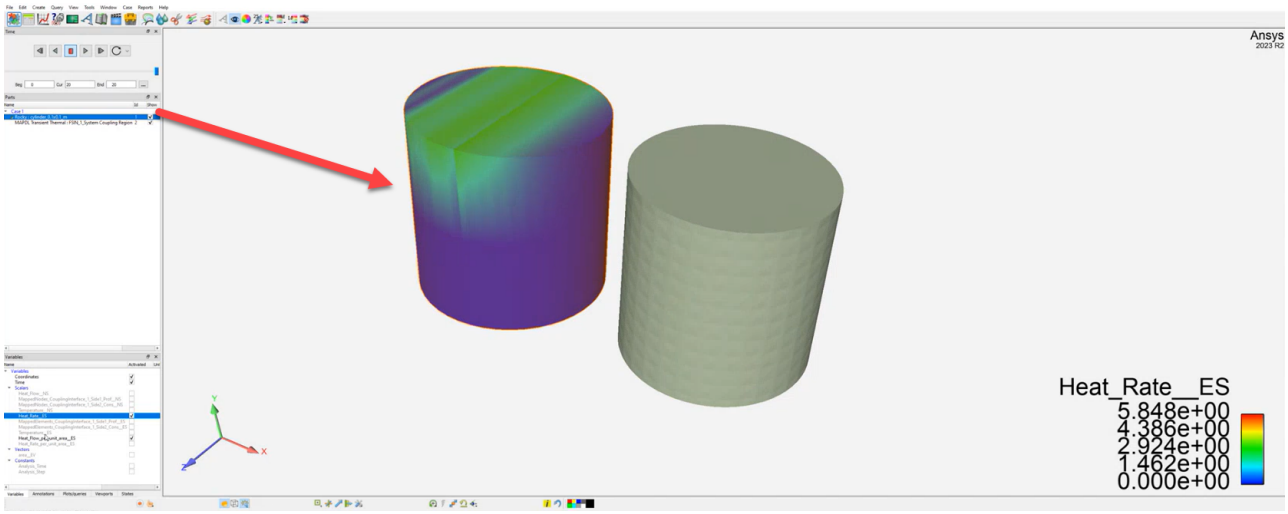


Figure 2.111: Geometries Solved



Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

Figure 2.112: Parameter Solved added to the geometry



See Also:

- Understand Rocky and Ansys Integration (p. 143)
- About Collecting and then Exporting Geometry Forces Data for Later FEM Analysis (p. 1159)
- About Rocky and Ansys Workbench Integration (p. 144)

2-WAY STRUCTURAL COUPLING WITH MECHANICAL

[HOW-TO VIDEO] Ansys Rocky: 2-Way Structural Coupling with Mechanical: https://www.youtube.com/watch?v=e89dBSdWarM&list=PL0lZXwHtV6Omiv62KRuPbnZ4oBK8BmDit&index=12

Ansys Products needed: Ansys Rocky, Ansys Mechanical, Ansys Workbench, System Coupling and Ensight.

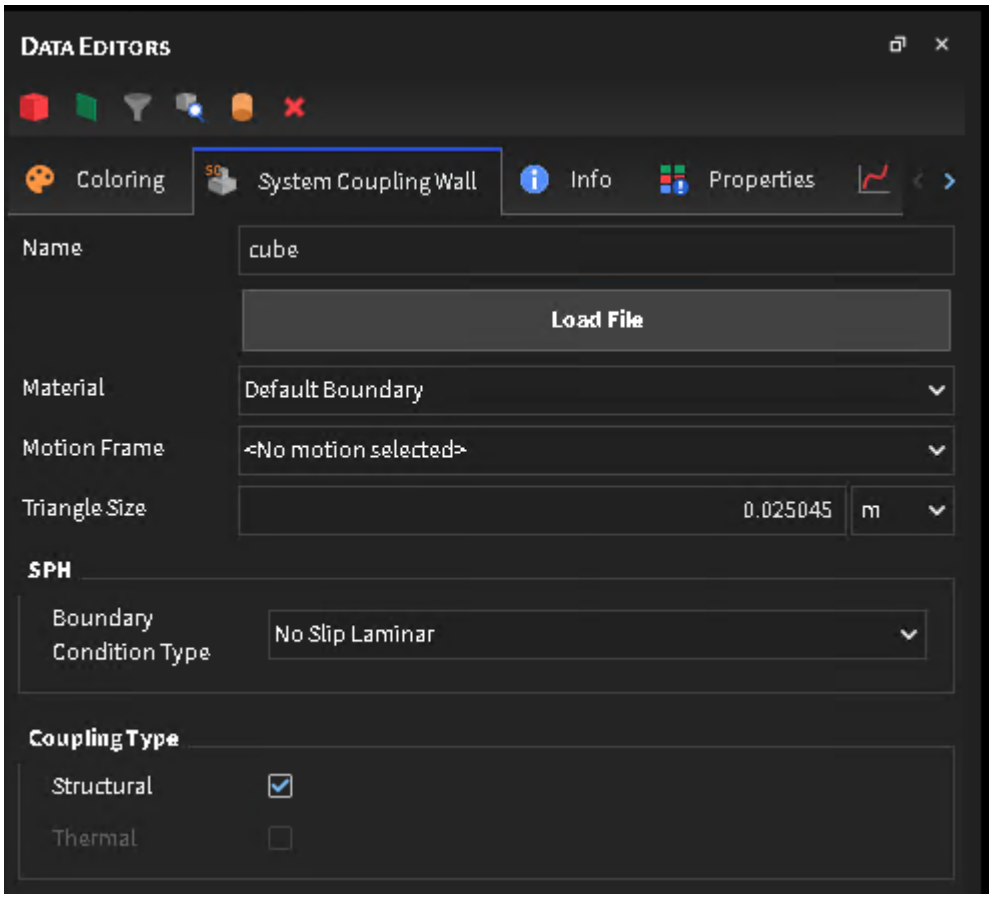
With this coupling, it is possible to send forces from Rocky and receive back the boundary displacement from Mechanical using the System Coupling.As the 2-Way Thermal coupling between Rocky and Mechanical (using the Transient Thermal one), the 2-Way Structural coupling uses the Transient Structural System inside Workbench. The main setup process is the same as already explained for the Thermal one, with just simple differences, that will be shown below.

1- SETUP IN ANSYS ROCKY

1. For the Rocky side, the first difference is that now, there are 2 Coupling Types options for the System Coupling Wall. For the 2-Way Structural Coupling, just select the Structural one. And for the 2-Way Thermal, just select the Thermal one.

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

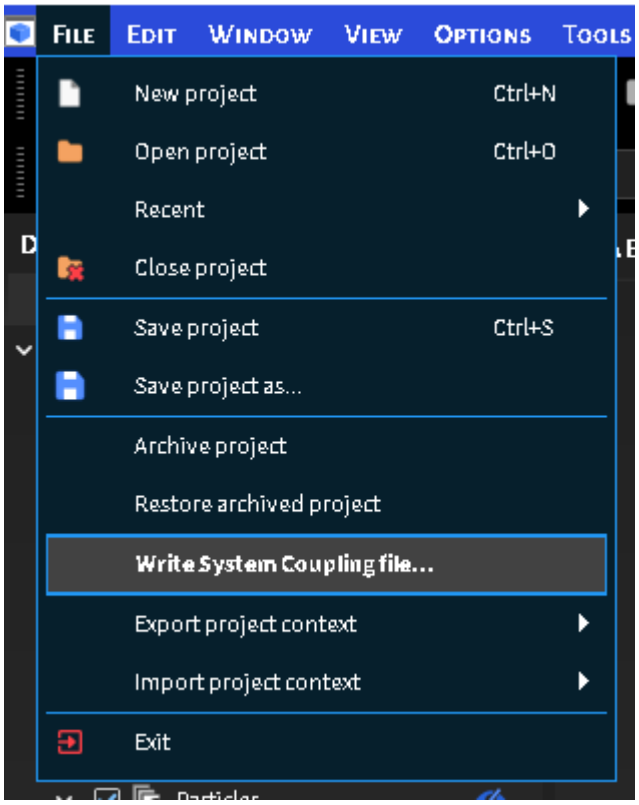
Figure 2.113: Coupling Types for the System Coupling Wall



2. The rest of Rocky setup, as well the Write System Coupling file is the same as 2-Way Thermal.

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

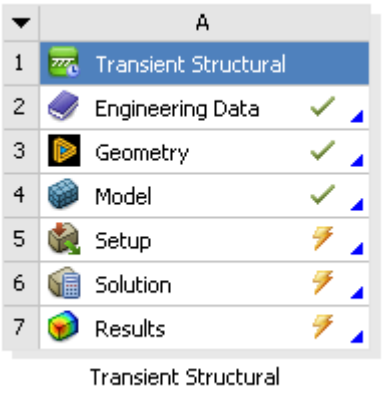
Figure 2.114: Write System Coupling File Rocky



2- SETUP IN ANSYS MECHANICAL

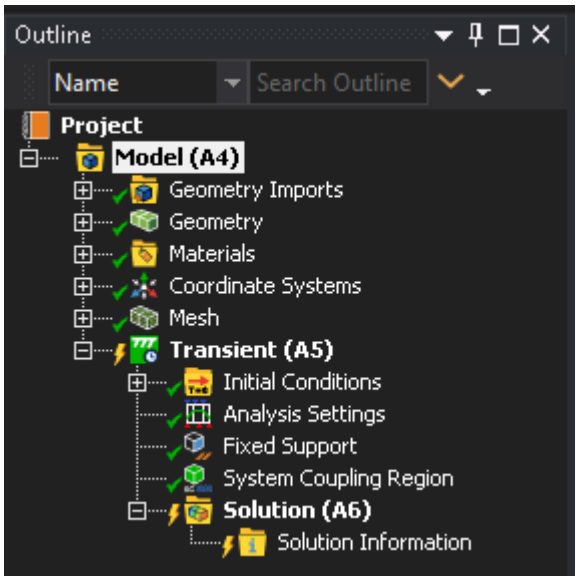
1. The difference from the Mechanical side is that now, the Transient Structural system for the Workbench that will be needed, as show in this image

Figure 2.115: Transient Structural Ansys Mechanical



2. The Model data tree setup process follows the same default Structural one, with just the System coupling regions as already described for the Thermal.

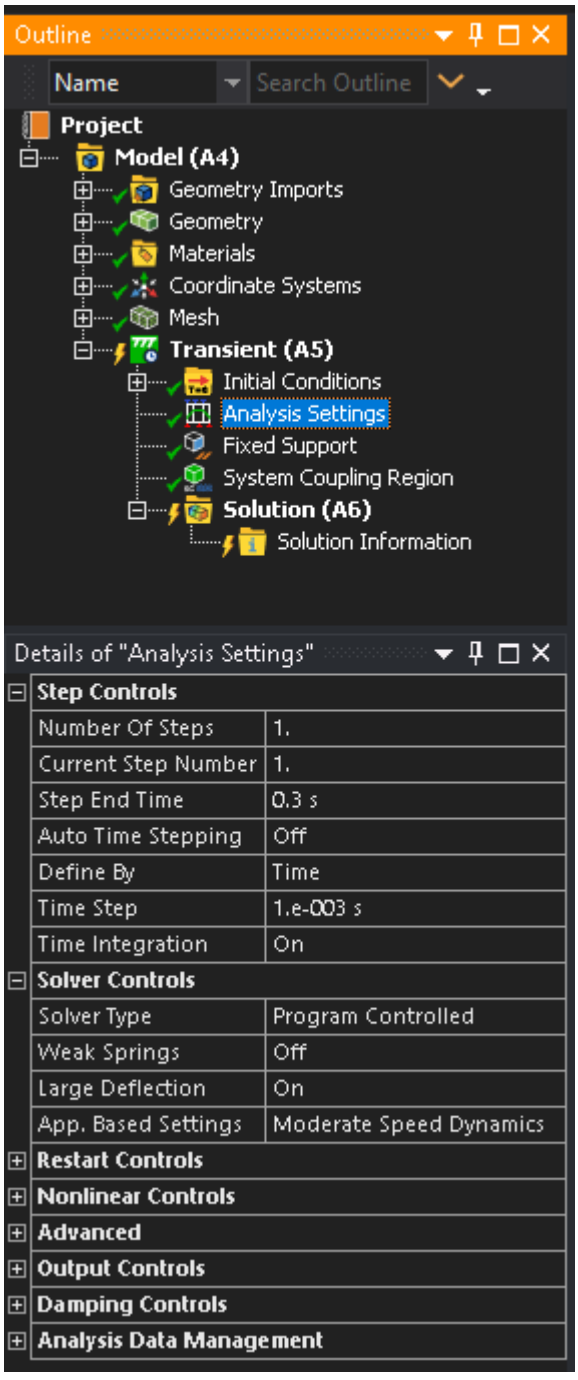
Figure 2.116: Project data tree for the Transient Strctural



3. One important point here is that the Time Step inside the Analysis System, for most of the cases, will need to have a small number, as the DEM-SPH collisions will happen in a small-time fraction, and as Mechanical is an Implicit Solver, the Time Step here will need to be smaller. With the biggest time steps, Mechanical will not be able to converge, and a High Element Distortion message will be shown in the System Coupling UI.

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

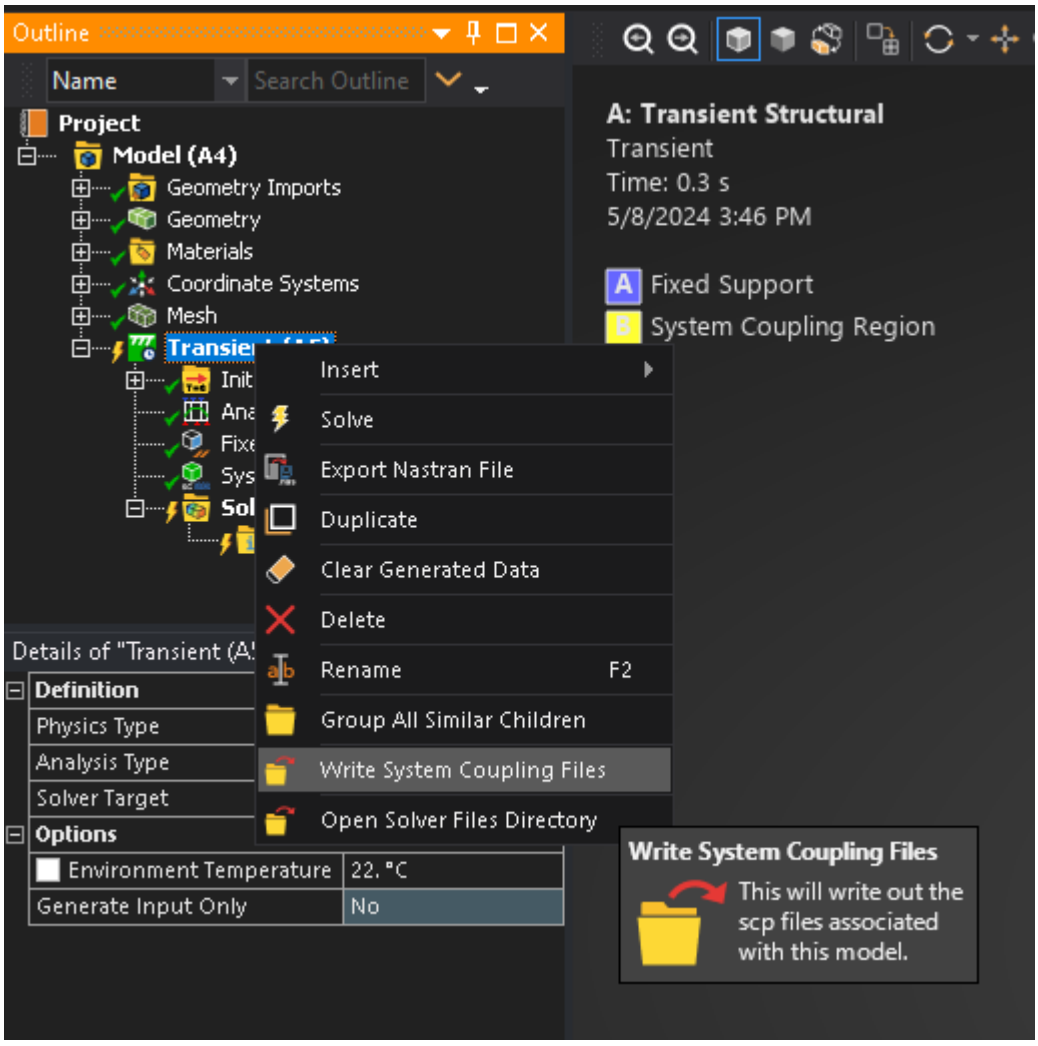
Figure 2.117: Time Step in Analysis Settings for the Transient Structural



1. After the setup, the System Coupling Region must be exported.

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

Figure 2.118: Write System Coupling Files



4- SETUP SYSTEM COUPLING

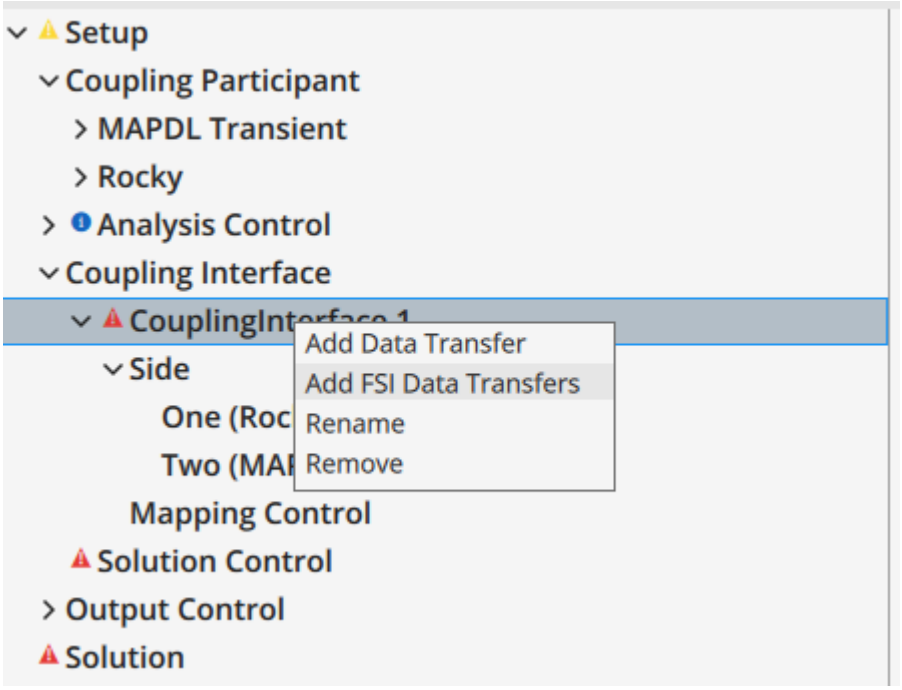
1. For the System Coupling side, all first steps are still the same. The only difference will be at Add Data Transfer, which now will have the Add FSI Data Transfer, which mainly will connect the forces data from Rocky, and send it to Mechanical.

Tip:

When you are doing this coupling, it is advisable to create three folders for your project, one for **Rocky**, another for **Workbench** and another for **System Coupling**.

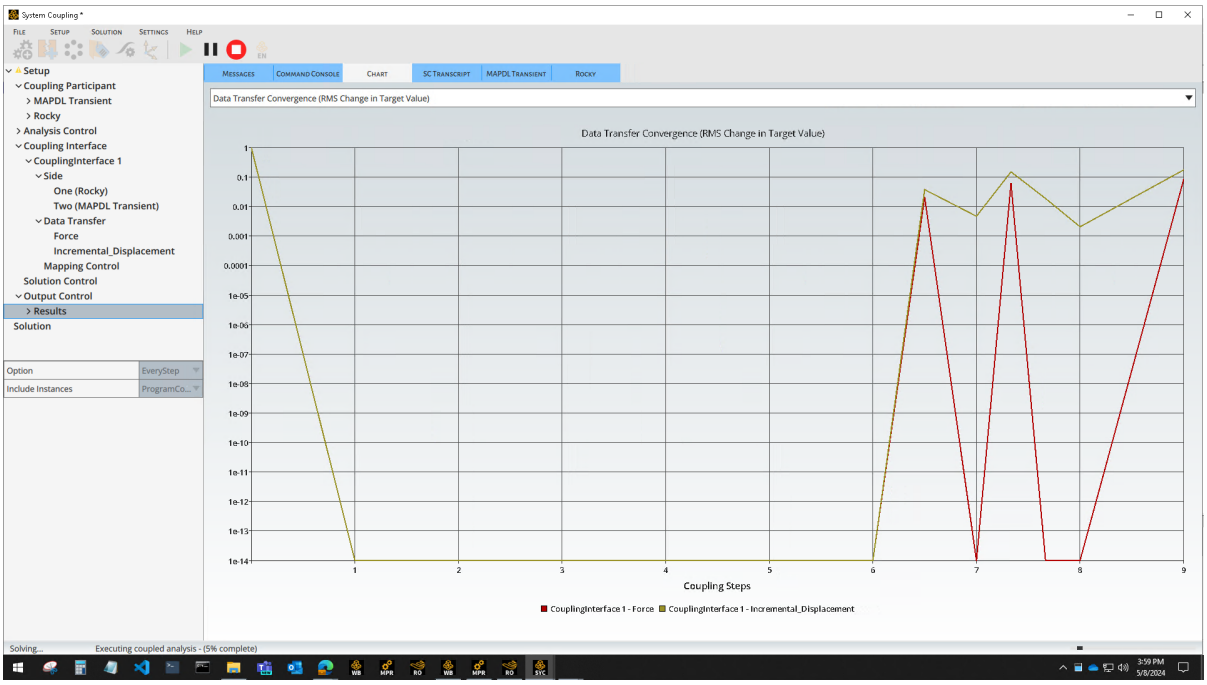
Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

Figure 2.119: Add FSI Data Transfers inside System Coupling



2. During the System Coupling solve process, it will be noticed that now, the Data Transfer will be the Force (Rocky to Mechanical) and Incremental Displacement (Mechanical to Rocky).

Figure 2.120: Force and Incremental Displacement Data Transfers



3. In the latest part, which is the post-processing, the two options remain: use Ensight directly from the System Coupling UI, or open Rocky and analyze the DEM-SPH trajectory and the Displacement results. At Ensight, new Variables are available, such as Force, Incremental Dis-

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

placement, and Displacement Since Mesh Import, and for the Rocky side, the Displacement property will show up.

Figure 2.121: Ansys Ensight Post-Processing

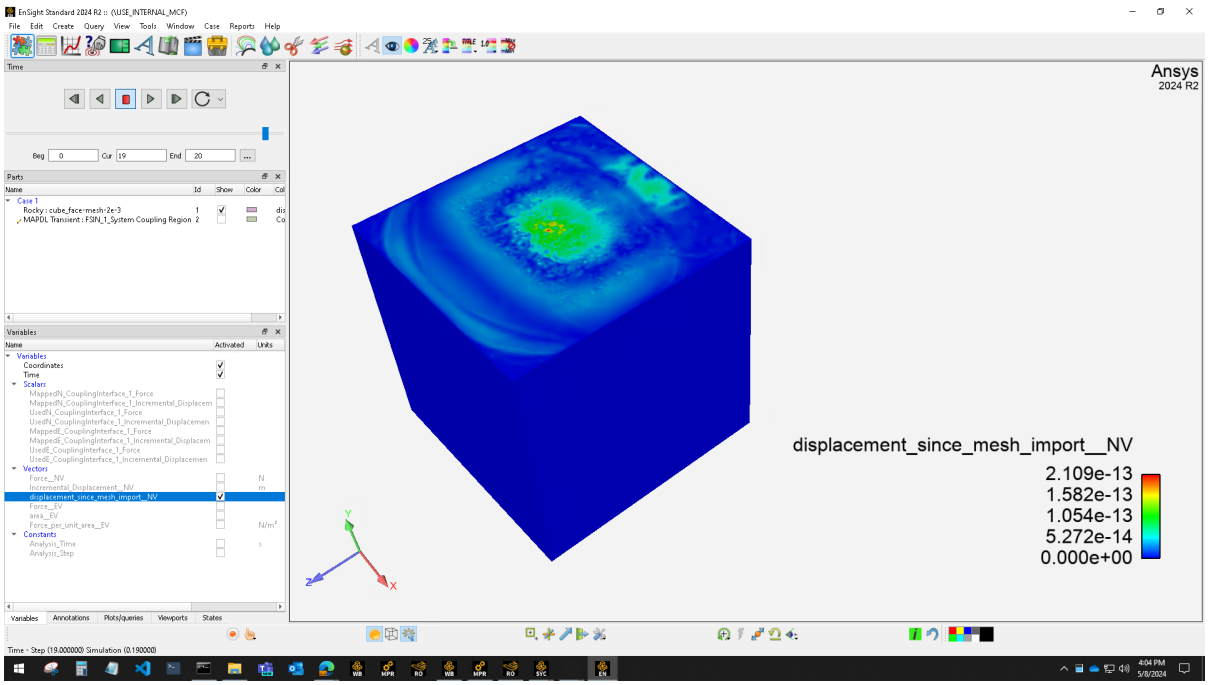
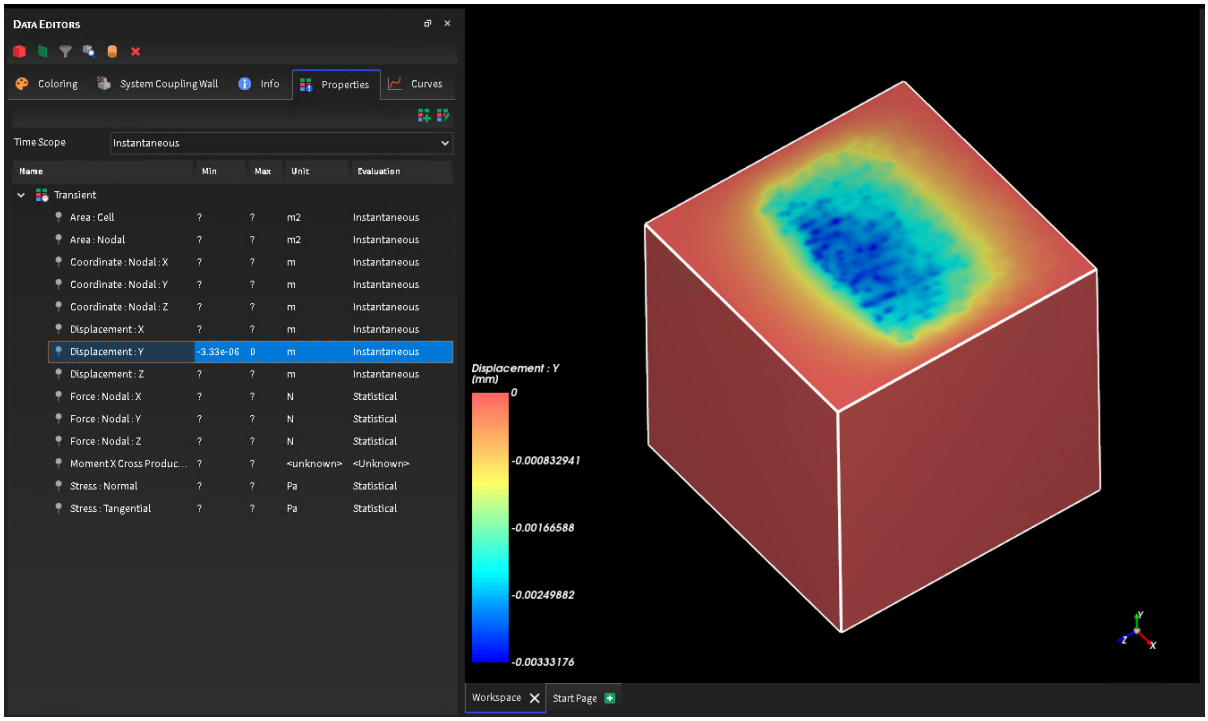


Figure 2.122: Ansys Rocky Post-Processing



See Also:

- Understand Rocky and Ansys Integration (p. 143)

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

- About Collecting and then Exporting Geometry Forces Data for Later FEM Analysis (p. 1159)
- About Rocky and Ansys Workbench Integration (p. 144)

2.3.3. Ansys Rocky and Ansys Fluent

Rocky's integration with Ansys Fluent in a 1-way coupled scenario enables Rocky to accept fluid flow and/or thermal property data from Fluent and then use that data to predict the resulting particle flow and/or related thermal properties. In a 2-way coupled scenario, Fluent and Rocky exchange fluid flow, particle forces, and thermal property data on a continual basis to discover how each interaction affects the other.

These two tools can operate as standalone products (for either 1-way or 2-way coupling) or can operate as connected components within Ansys Workbench (for 1-way coupling). But in order to make use of Fluent, an Ansys Coupling Component must be installed. (See also Install Ansys Coupling Components (p. 792).)

This section covers the following topics:

IROCKY AND FLUENT BOUNDARY THERMAL COUPLING (p. 197)

THERMAL COUPLING REQUIREMENTS (p. 198)

2-WAY COUPLING (p. 199)

1-WAY COUPLING (p. 203)

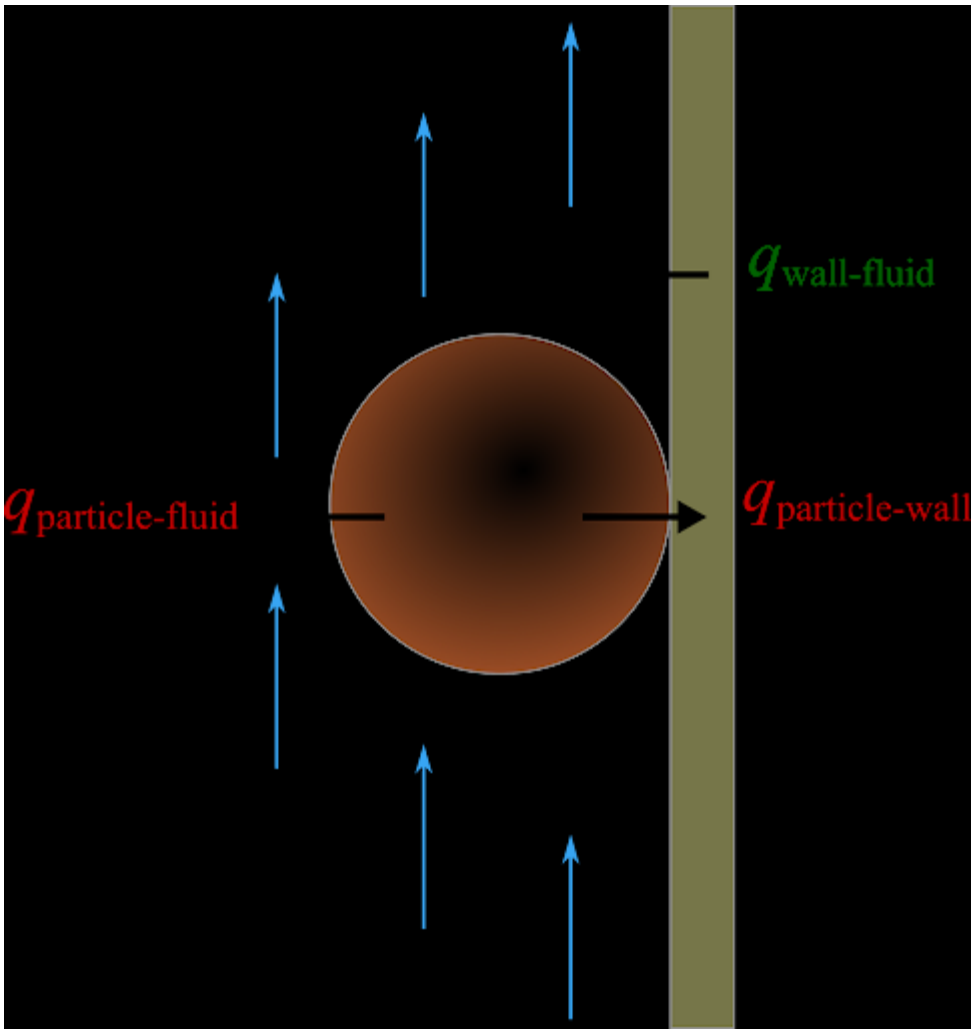
ROCKY AND FLUENT BOUNDARY THERMAL COUPLING

Rocky 23R1 allows the computation of the heat transfer between particles and walls. For 2-way unresolved simulations, the heat exchange during collisions is transferred to the CFD solver and used to update the surface temperature. That way, the temperature of each boundary triangle is not predefined in Rocky, but is imported from the CFD solver at every time step.

- For both 1-way and 2-way coupled simulations, the particle temperature is computed by Rocky accounting for the conductive heat transfer during collisions against walls and against other particles, as well as for the convective heat exchange with the fluid phase.
- For 2-way coupled simulations, the boundary temperature changes are computed by the CFD solver accounting for the heat transferred to/from the fluid phase and the heat exchanged with the particles during particle-wall collisions.
- For 1-way coupled simulations, the wall temperature changes according to the CFD solution exported to Rocky (and can vary spatially and in time), but the heat exchanged during particle-wall collisions is not transferred to the CFD solver.

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

Figure 2.123: Thermal coupled simulation schematics.



The coupled boundaries are automatically imported from the Fluent setup by selecting the desired walls from a list of compatible walls automatically populated in the CFD Coupling Boundaries tab.

THERMAL COUPLING REQUIREMENTS

These are the requirements for setting up a thermally coupled simulation.

1. The Rocky setup must have the **Thermal model** enabled.
2. The Fluent setup must have the **Energy Equation** turned on.
3. For 2-way coupled simulations, the walls must have **Shell Conduction** enabled in the Fluent setup, regardless of the Thermal Conditions option chosen.
4. Giving the wall thickness (**thin wall** model) but not turning on the **shell conduction** model does not make the wall compatible with the coupling.
5. Only walls in participating fluid domains can be thermally coupled to the DEM solution.

CURRENT LIMITATIONS

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

- Mesh motion is not supported for thermally coupled boundaries.
- Dynamic meshes (deforming meshes and remeshing) are not supported.
- Polyhedral meshes are not supported.

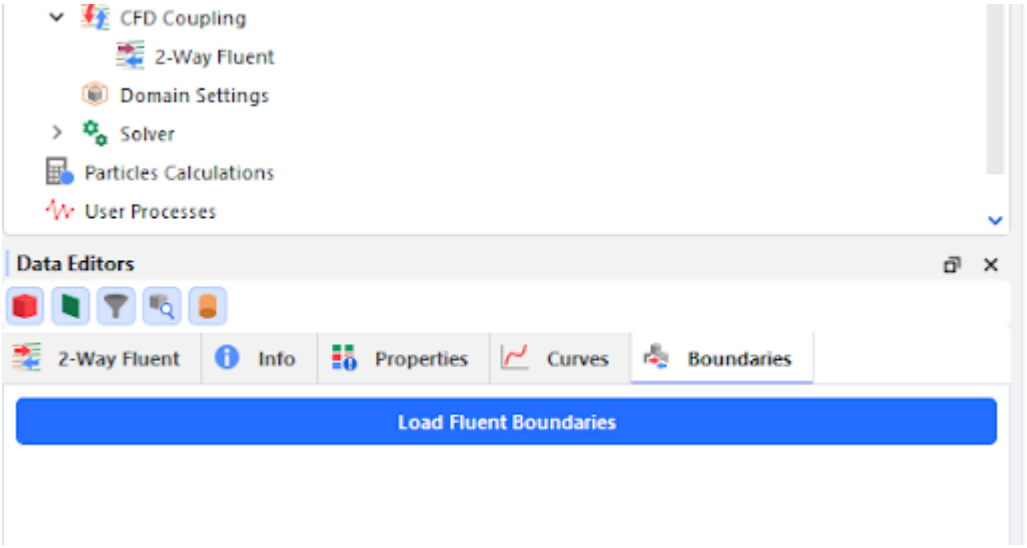
To set up a **Thermal Coupled Simulation**, follow the steps below:

2-WAY COUPLING

1. Import the Fluent case file and set up the required models listed in the **Interactions, Coupling, Zones and Interfaces, Fluent** and **Variables** tabs, as required for any 2-way coupled simulation.
2. Click the **2-Way Fluent** entry in the Data Panel and navigate to the **Boundaries** tab in the Data Editors. Clicking on **Load Fluent Boundaries** will open a window listing all the compatible boundaries.

   Tip: Check the Requirements (p. 198) entry to understand the compatibility criteria.

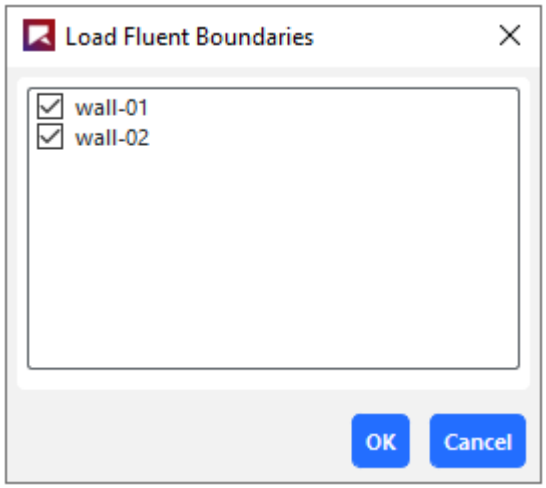
   Figure 2.124: 2-way Fluent Boundaries tab in the Data Editors .



3. Select the walls you want to use for the thermal coupling.

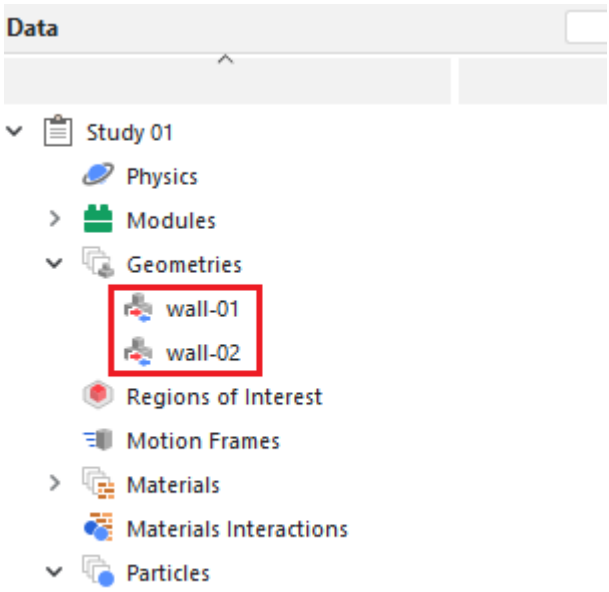
Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

Figure 2.125: Select the walls in the Load Fluent Boundaries.



4. For each selected wall, a new entry will appear under **Geometries** in the Data panel. These walls have specific icons as they are thermally coupled.

Figure 2.126: New entries under Geometries in the Data panel.



5. Unselecting the wall from the **Load Fluent Boundaries** list will remove the coupled wall from the **Geometry** list, as well as any user process linked to the coupled wall.

6. The coupled walls have fewer options compared to the standard walls as some models are not supported, such as wear and replication.

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

Figure 2.127: Options available for a coupled wall.

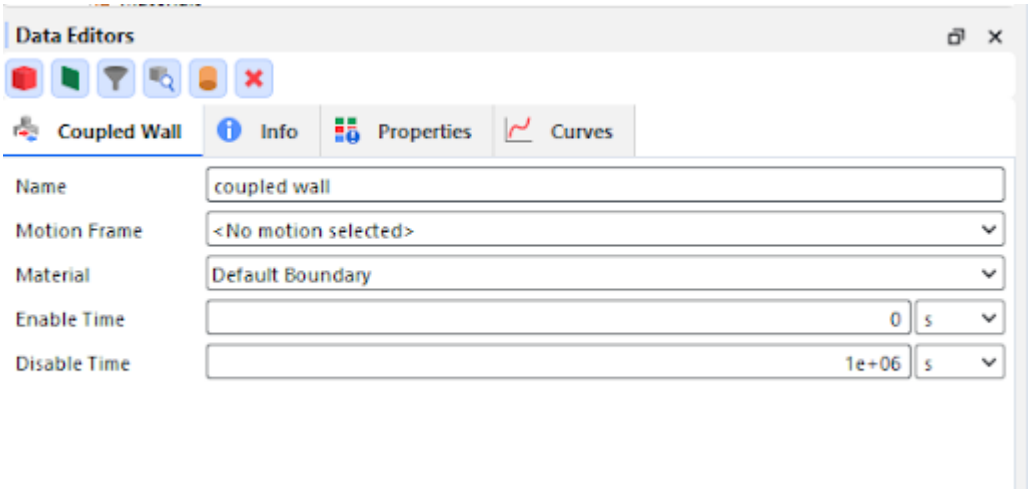
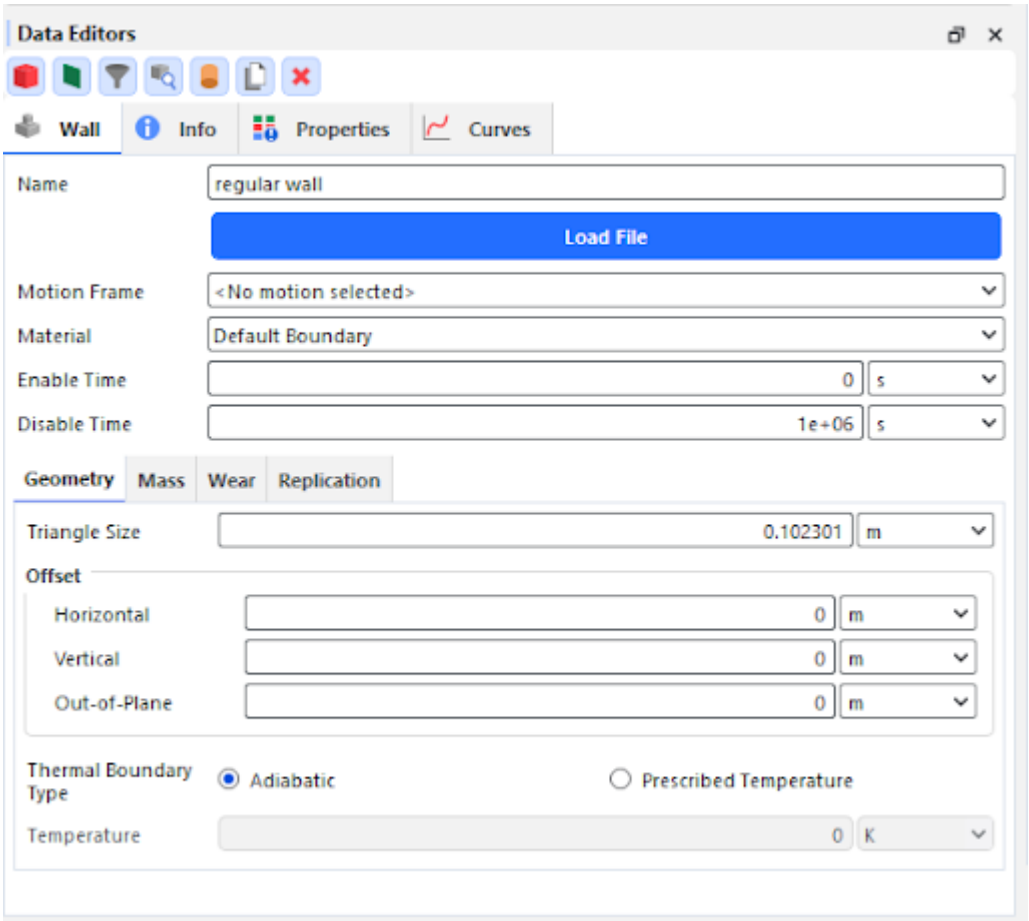


Figure 2.128: Options available for a standard wall.



7. Select the **Material** to be assigned to the coupled wall.

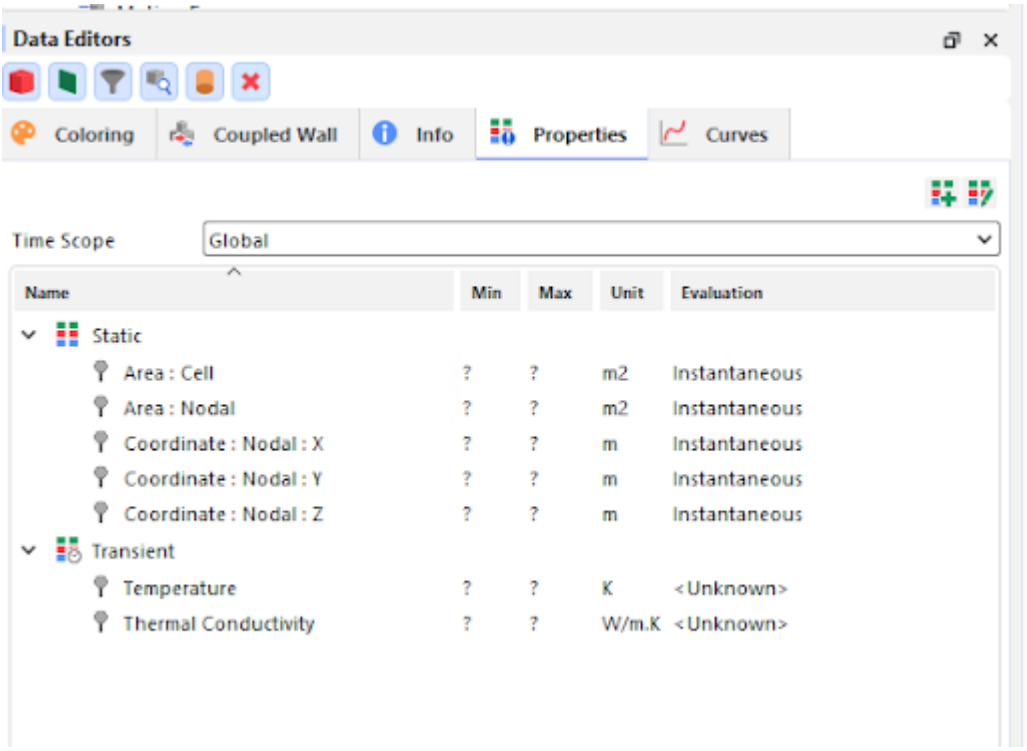
   Important: The mechanical properties of the assigned material will be used for computing contact forces for collisions between the particles and the wall, but the thermal properties required for the conductive heat transfer calculation will be extracted from Fluent.

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

8. Run the coupled simulation normally.

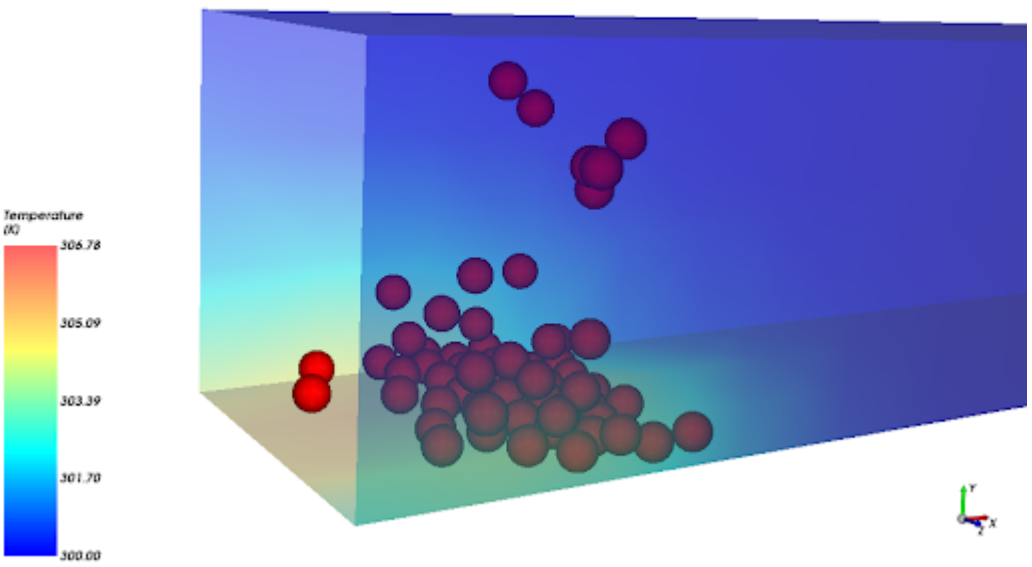
9. Two additional transient properties are available for coupled walls - **Temperature** and **Thermal Conductivity**. These triangle properties can be shown in 3D views or used to generate quantitative analyses such as time plots, histograms, or output parameters.

Figure 2.129: Additional transient properties for coupled walls.



Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

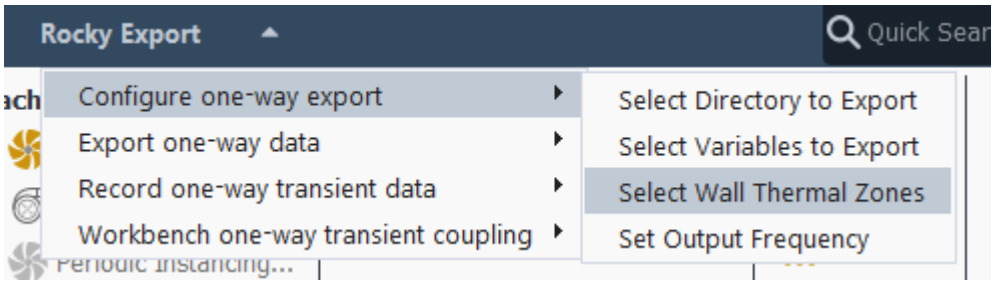
Figure 2.130: The wall temperature, initially at 300K, increases as hot particles (500K) collide with the wall.



1-WAY COUPLING

1. For setting up a 1-way thermal coupled simulation, there is only one additional step that must be completed. In the **Rocky Export** plugin, select **Configure one-way export** and then **Select Wall Thermal Zones**.

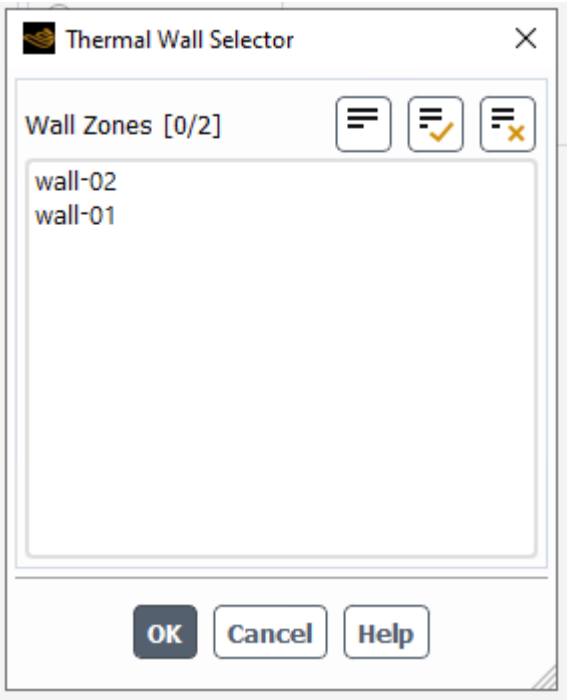
   Figure 2.131: Select Wall Thermal Zones in the Rocky Export plugin.



2. The **Thermal Wall Selector** window will list all the walls that are compatible with the 1-way thermal coupling between Rocky and Fluent.

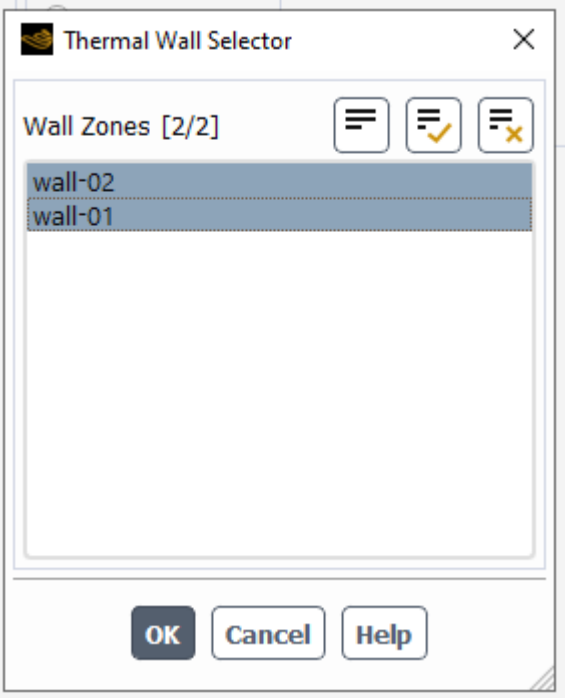
Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

Figure 2.132: List of all the walls compatible with the 1-way thermal coupling between Rocky and Fluent in the Thermal Wall Selector window.



3. Pick the walls that you wish to participate as coupled walls.

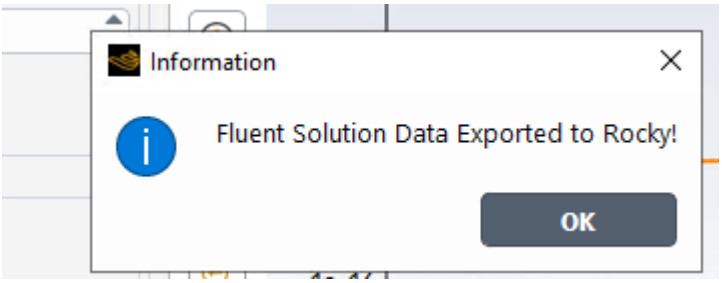
Figure 2.133: Walls selected in the Thermal Wall Selector window.



4. Run the simulation normally and export the results to Rocky using the Rocky Export Plugin.

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

Figure 2.134: Results exported to Rocky using the Rocky Export Plugin.



5. In Rocky, enable Thermal and select Fluent (Fluid -> Particle) as the coupling mode.

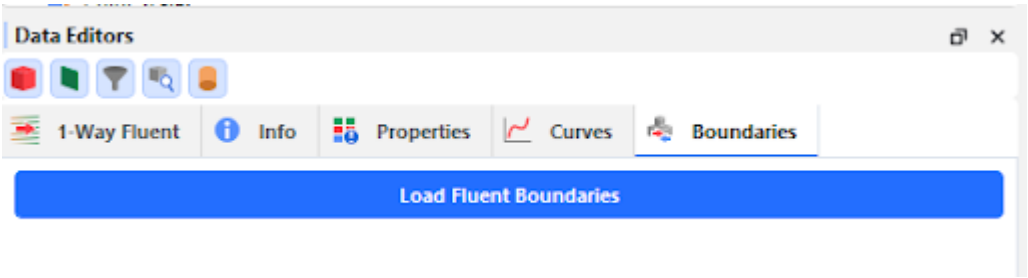
Figure 2.135: Fluent (Fluid -> Particle) Selected.



6. Select the F2R file that was exported from Fluent. Set up the required models listed in the **Interactions** and **Coupling** tabs, as required for any 1-way coupled simulation.

7. Click the 1-way Fluent entry in the Data Panel and navigate to the **Boundaries** tab in the Data Editors.

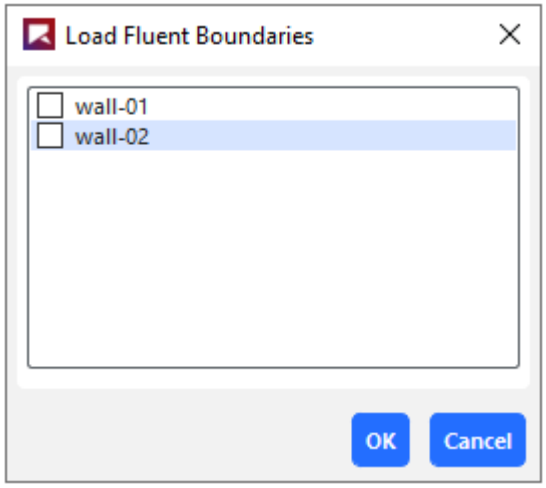
Figure 2.136: 1-way Fluent Boundaries tab in the Data Editors.



8. Clicking on **Load Fluent Boundaries** will open another window listing all the compatible boundaries. **Tip:** Check the Requirements (p. 198) entry to understand the compatibility criteria.

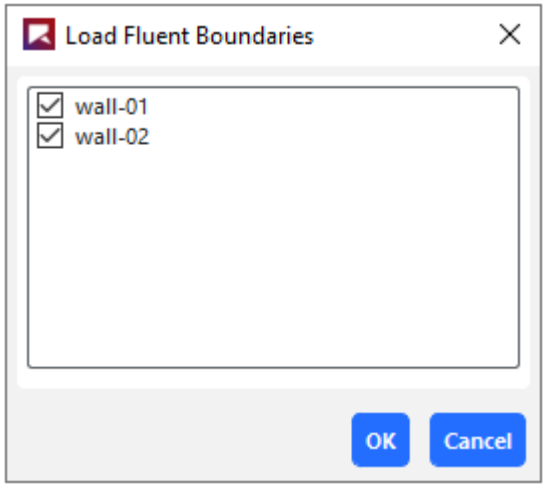
Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

Figure 2.137: Load Fluent Boundaries listing all the compatible boundaries.



9. Select the walls you want to use for the thermal coupling.

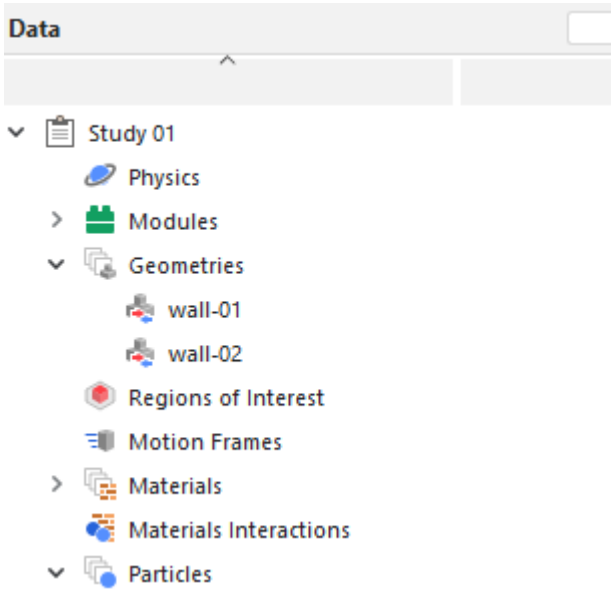
Figure 2.138: Walls to be selected for the thermal coupling.



10. For each selected wall, a new entry will appear under **Geometries** in the Data panel. These walls have specific icons as they are thermally coupled.

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

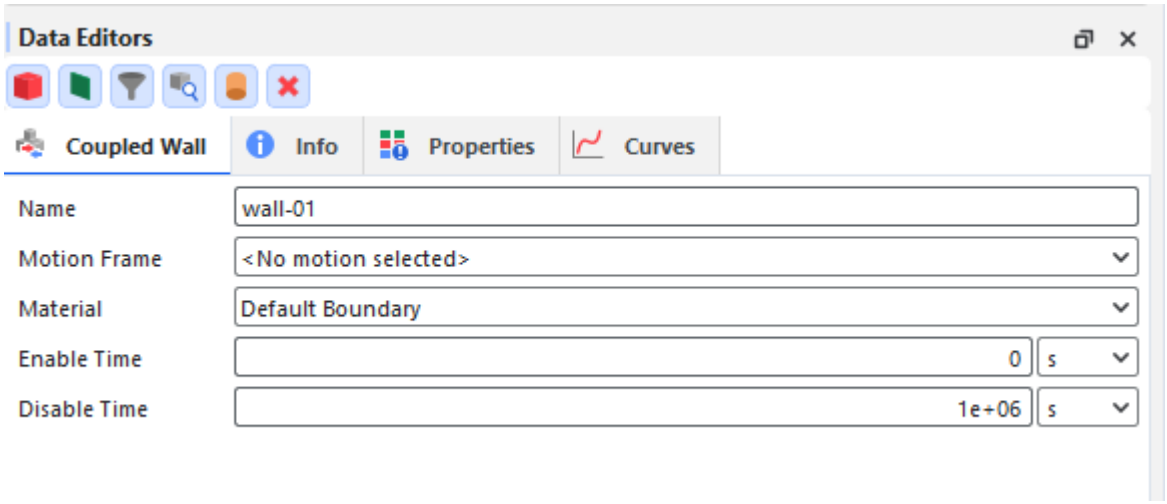
Figure 2.139: New entries under Geometries in the Data panel.



11. Unselecting the wall from the **Load Fluent Boundaries** list will remove the coupled wall from the **Geometry** list, as well as any user process linked to the coupled wall.

12. The coupled walls have fewer options compared to the standard walls as some models are not supported, such as wear and replication.

    Figure 2.140: Options available in Coupled Walls.



13. Select the **Material** to be assigned to the coupled wall.

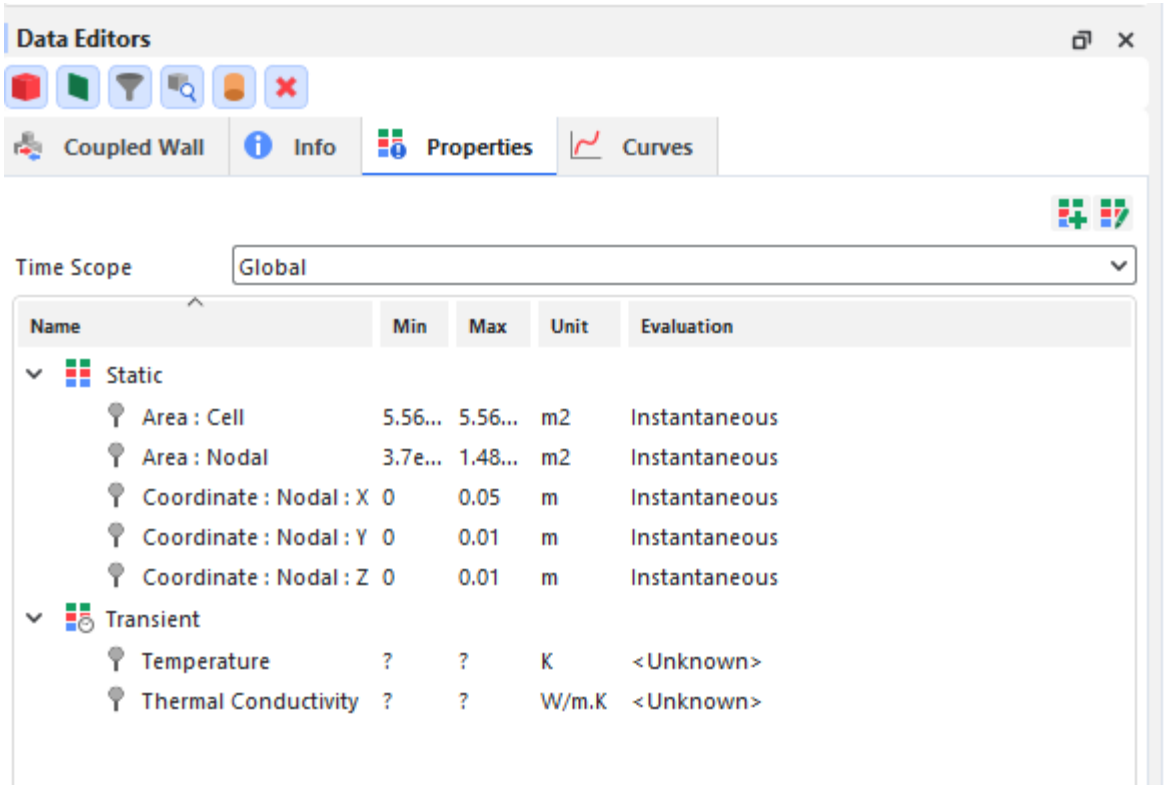
    Important: The mechanical properties of the assigned material will be used for computing contact forces for collisions between the particles and the wall, but the thermal properties required for the conductive heat transfer calculation will be extracted from Fluent.

14. Run the coupled simulation normally.

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

15. Additional properties are created for the coupled walls. These can be used for quantitative analyses, such as time plots and histograms, for creating output parameters or for coloring the wall triangles in a 3D view.

Figure 2.141: Properties created for the coupled walls.



The ways in which Rocky and Ansys Fluent can work together are covered in the following topics:

- About Using the 1-Way Fluent Method (p. 775)
- Set Up and Run a 1-Way Fluent Transient Project within Ansys Workbench (p. 775)
- About Using the 2-Way Fluent Method (p. 795)
- About Rocky and Ansys Workbench Integration (p. 144)

See Also:

- Understand Rocky and Ansys Integration (p. 143)

2.3.4. Ansys Rocky and Ansys Motion

In version 2025 R1 of Rocky, the coupling with Ansys Motion was included into an embedded module. By doing this, the Rocky module for FMU Coupling from the Ansys Motion Coupling installer was removed. A built-in Rocky module called Multibody Dynamics FMU Coupling was created.

The Ansys Motion Coupling installer will only install the FMU Export extensions for Ansys Motion. To run a coupled simulation with any FMU file, the user must enable the Multibody Dynamics FMU

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

Coupling Rocky module. Another detail is that the user won't need to install the modules for each user on the same machine anymore.

For additional details regarding the Ansys Motion Coupling can be found in the Installation Guide.

See Also:

- Understand Rocky and Ansys Integration (p. 143)

2.3.5. Ansys Rocky and Ansys optiSLang

Improved integration between Ansys optiSLang and Ansys Rocky can boost your iterative design processes. By combining your Ansys Rocky cases with the robust design optimization (RDO) methods of optiSLang, processes that require multiple repeat runs - such as those for material calibration - become faster and more efficient. And when combined with the powerful parametric modeling capabilities of Ansys Workbench, you can take these efficiencies even farther.

Using the optiSLang plugin, the input parameter selection and variation for your Rocky projects are automatic based on sensitivity analyses and meta-modeling techniques. These automatic methods reduce the number of iterative cases that need to be run and increase the confidence with which you can adjust the interaction parameters for your calibration projects.

Important:

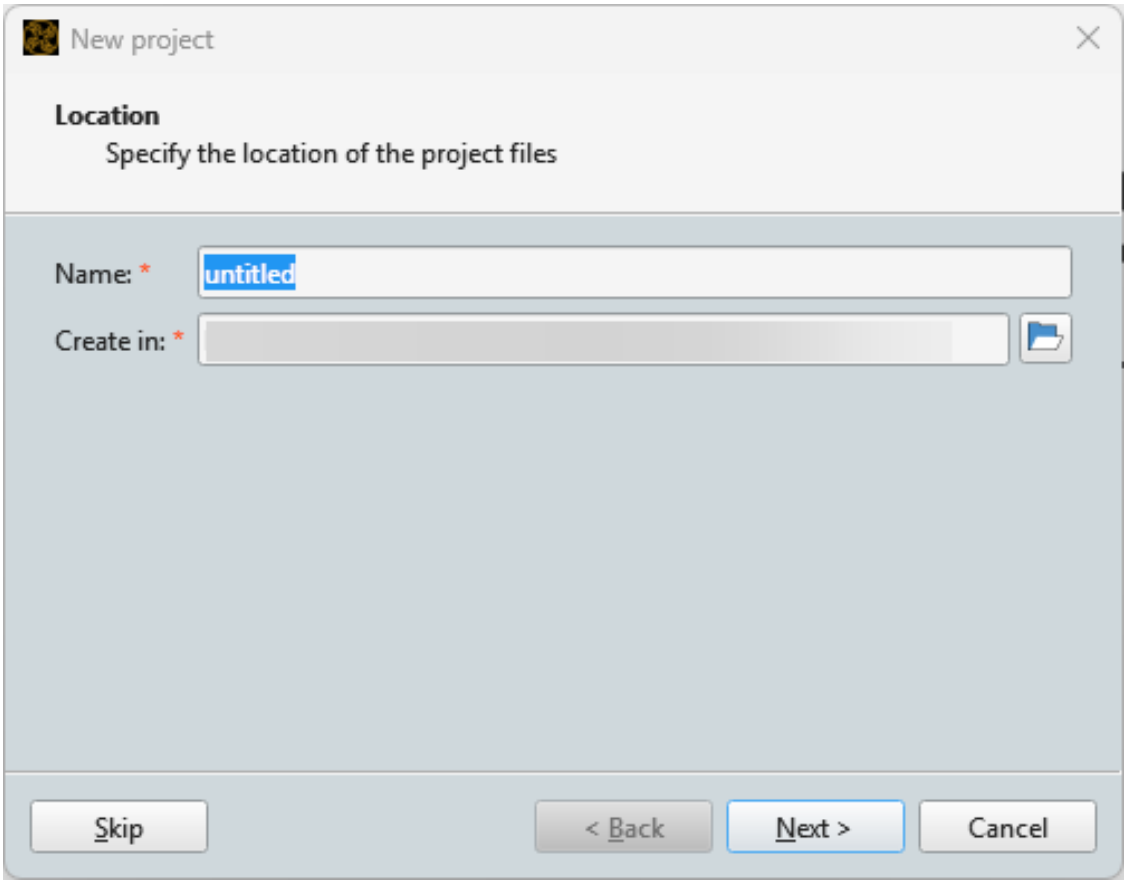
Before you begin, ensure that the Rocky project you want to integrate with optiSLang has already been processed in Rocky and has simulation results. If you have only one license of Rocky, ensure that your Rocky program is closed, as optiSLang software will need to open it as part of the integration process.

It is possible to integrate Ansys optiSLang with Ansys Rocky, as shown below:

1. Open the **Ansys optiSLang** program.
2. Create a new optiSLang project.

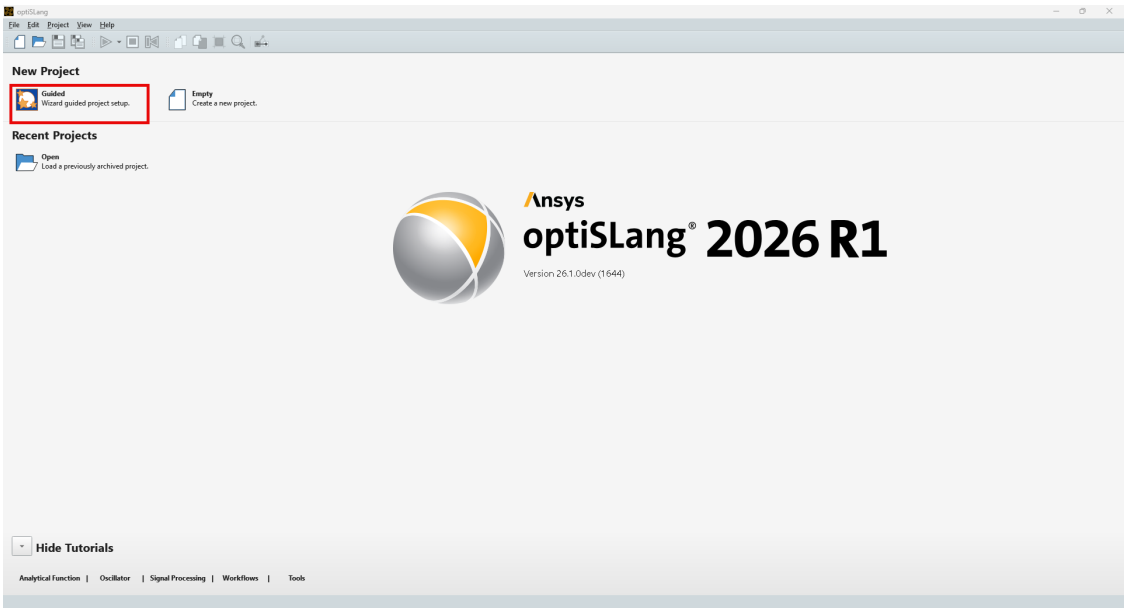
Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

Figure 2.142: New Project



3. From the main Ansys optiSLang screen, under **New Project**, click **Guided** (as shown) to open the Solver Wizard.

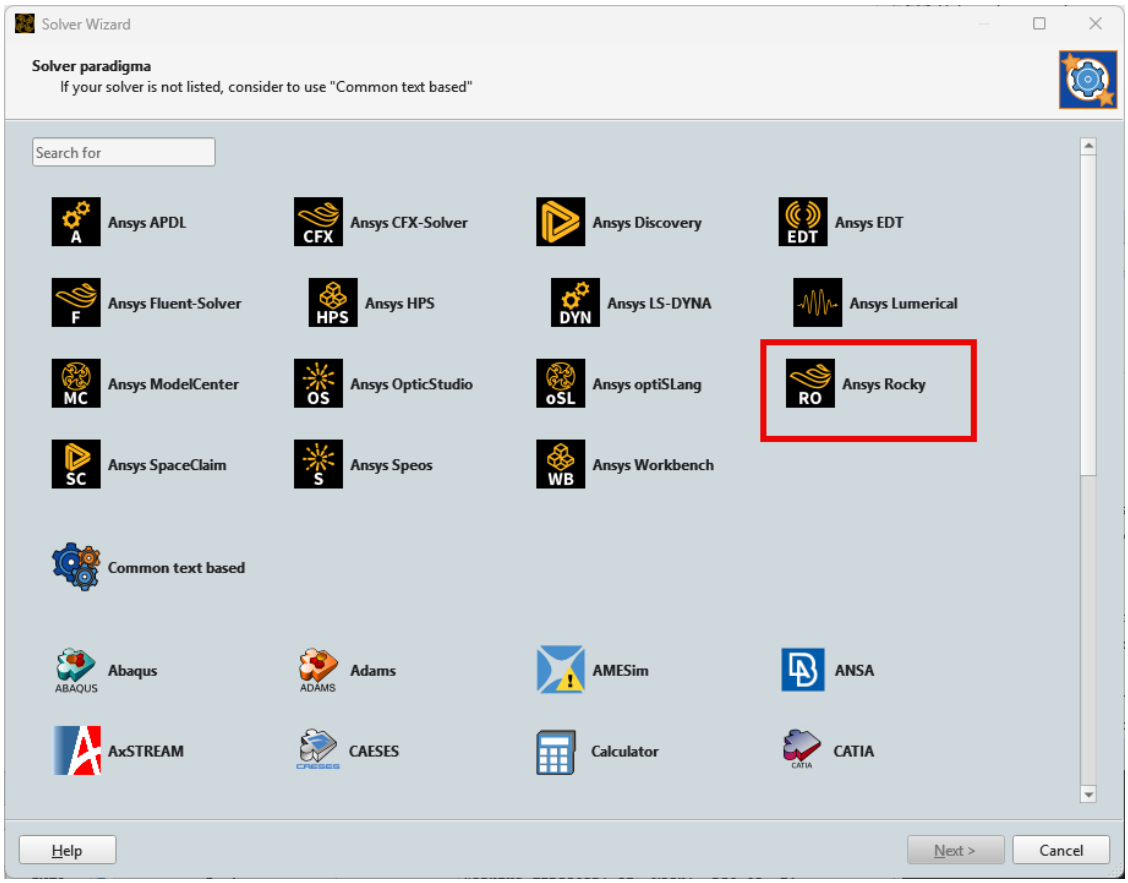
Figure 2.143: Solver Wizard



Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

4. From the Solver Wizard, search for Rocky, and then under Interfaces, select the Ansys Rocky integrated module as shown below:

Figure 2.144: Ansys Rocky selection in Ansys optiSLang software



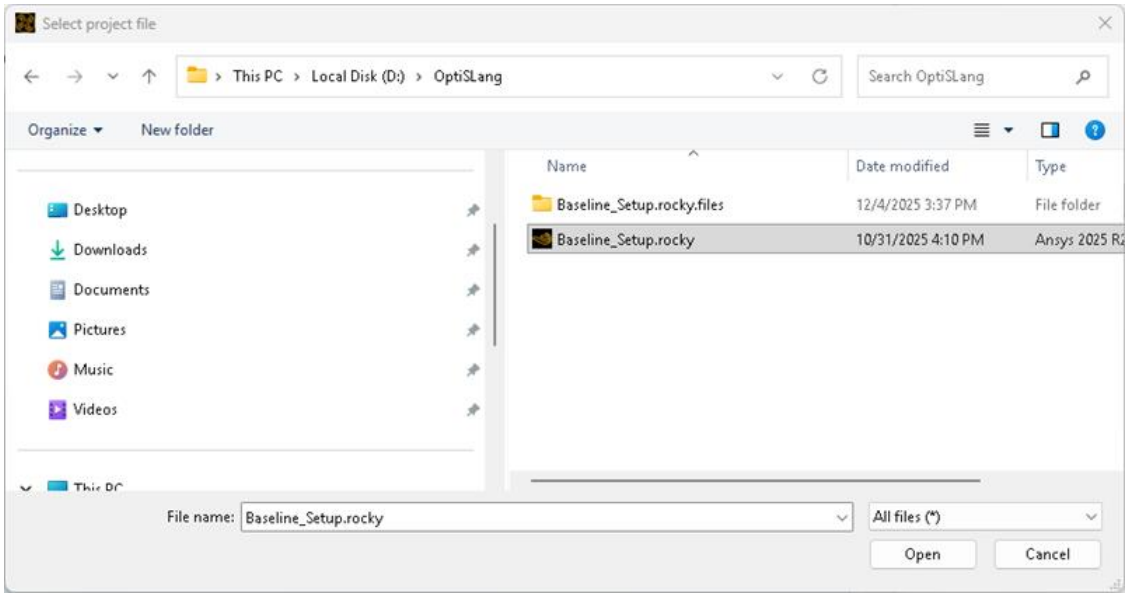
Important:

Starting with version 2026 R1, a new **BETA** *Integration Module* is available to access Ansys Rocky in optiSLang software, which brings improved usability. See about it in the section Ansys Rocky (Beta) Integrated Module in optiSLang software (p. 212)

5. From the **Select Project File** dialog, navigate to Ansys Rocky and then select the already-processed Rocky project file you want to use. Click Open (as shown).

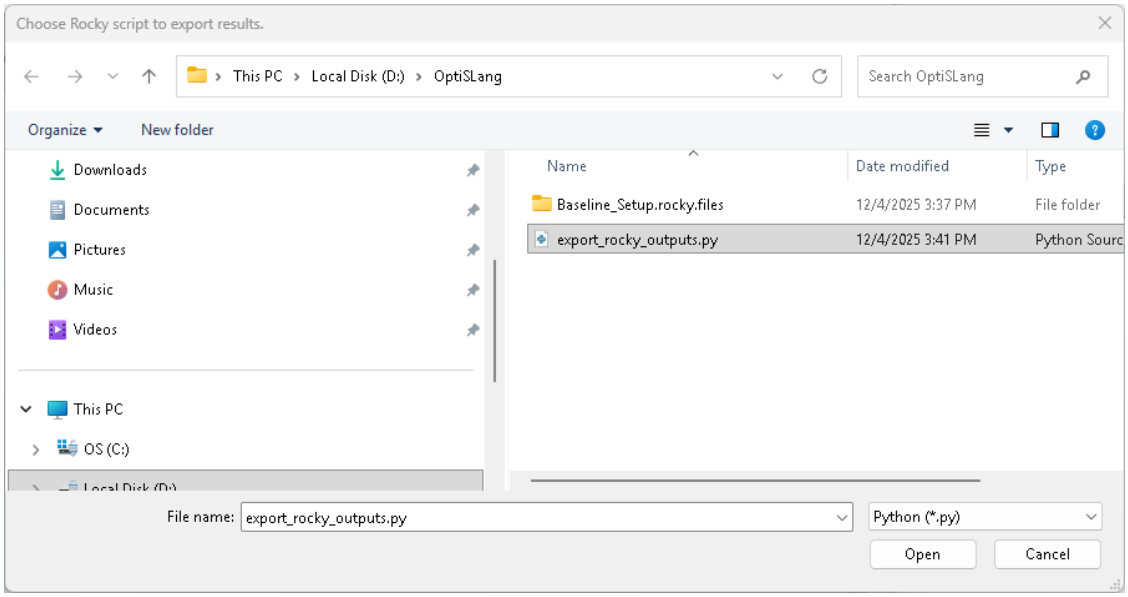
Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

Figure 2.145: Project file Selection



6. From the **Choose Rocky script to export results** dialog, select the script (.py) file you want optiSLang to use for the exported output variables, and then click **Open** (as shown).

Figure 2.146: Choose the Rocky script to export results



Note:

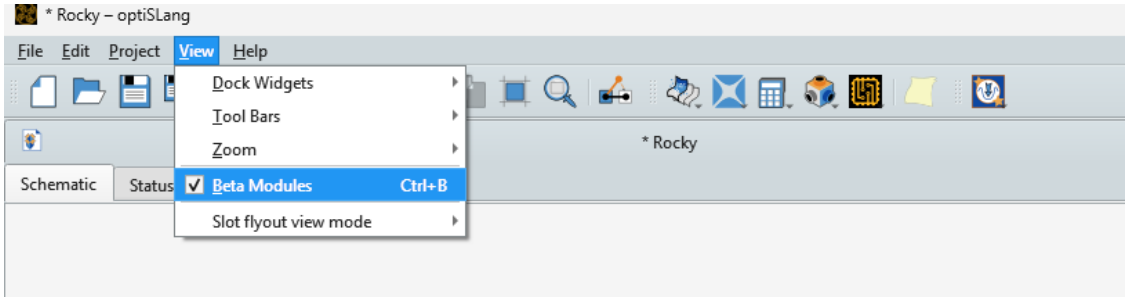
By default, optiSLang creates .py files for this purpose in the same folder as the Rocky project you selected earlier. After optiSLang loads the input/output variables, Rocky will launch and you will receive a confirmation message.

ANSYS ROCKY (BETA) INTEGRATED MODULE IN ANSYS OPTISLANG

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

Starting with version 2026 R1, it is also possible to use the new **Ansys Rocky (Beta)** integrated module to improve the usability of this coupling. To do this, simply activate the beta modules function as illustrated in the image below:

Figure 2.147: Enabling Beta Modules in Ansys optiSLang

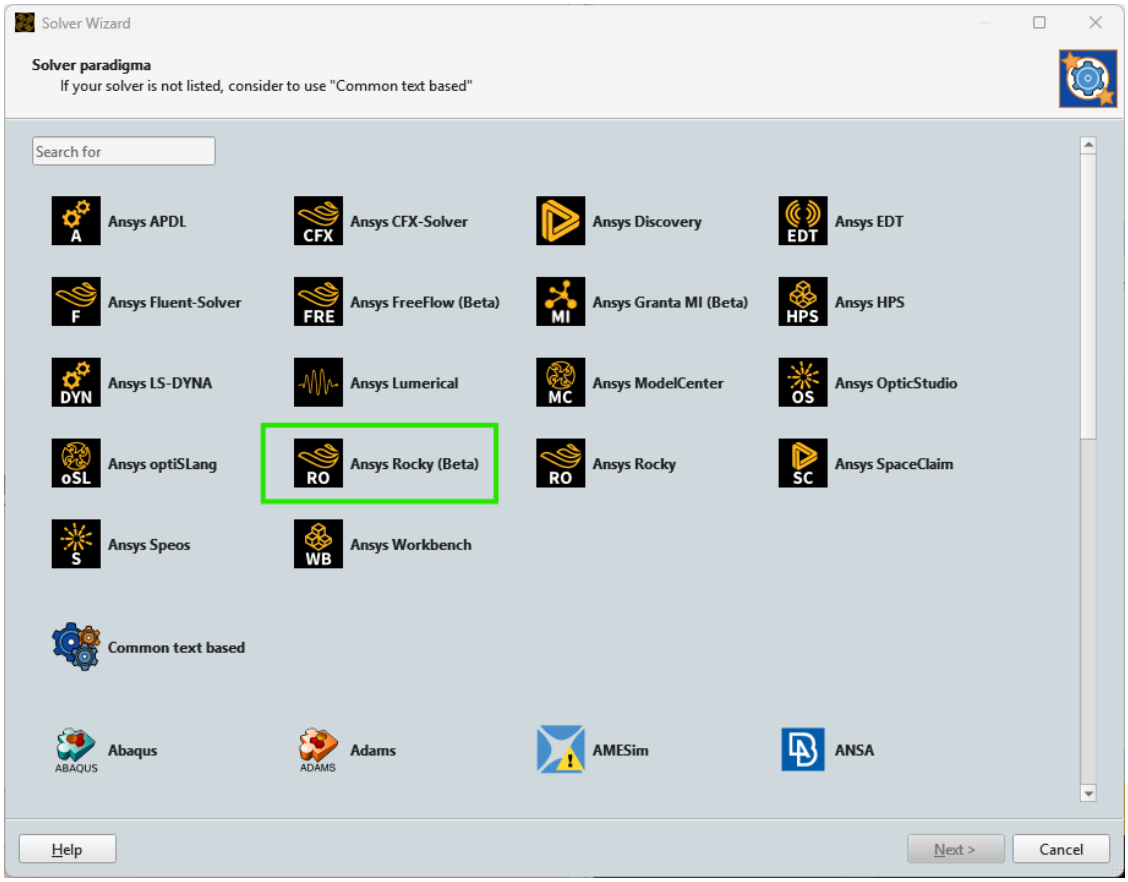


1. Then follow step 2 (p. 209) and step 3 (p. 210) explained above.

2. From the Solver Wizard, search for Rocky, and then under Interfaces, select the **Ansys Rocky (Beta)** integrated module as shown below:

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

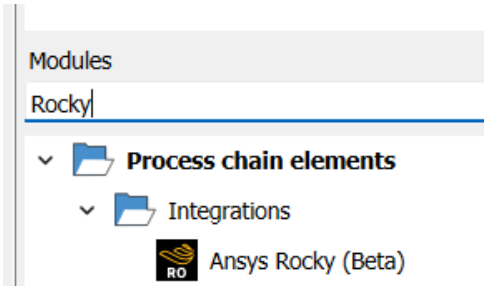
Figure 2.148: Ansys Rocky (Beta) selection in Ansys optiSLang software



Note:

Another way to use the Ansys Rocky (Beta) module is by adding it from the ***Modules*** dock -> ***Process chain elements*** -> Integrations, as shown below.

Figure 2.149: Ansys Rocky (Beta) selection in Modules Dock



3. After that, follow step 5 (p. 211) shown above to select your Rocky file.

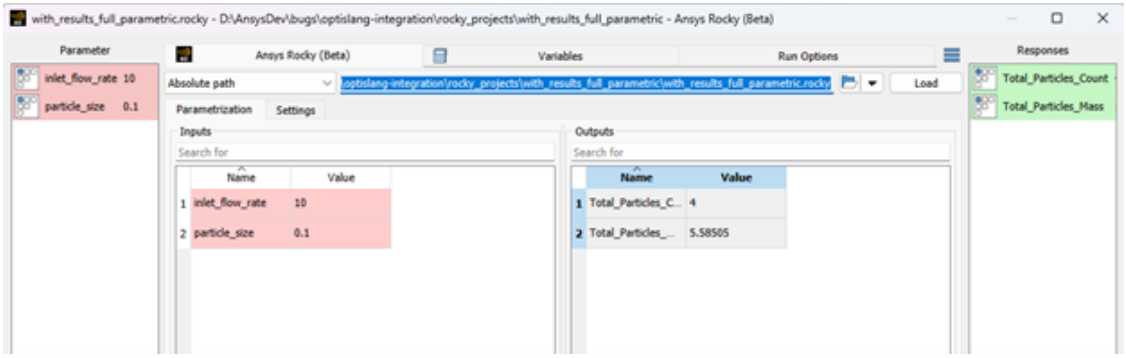
Note:

With this new integrated module, step 6 is not necessary.

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

4. In the ***Ansys Rocky (Beta) module window***, you will be able to define the solver settings, as shown in the Figure below. If you have loaded the Ansys Rocky (Beta) module from the ***Modules*** dock instead of using the Solver Wizard, you will be required to enter the Rocky project path.

Figure 2.150: Ansys Rocky (Beta) Module Solver Settings



If your Rocky project has Input parameters, they will be listed under the ***Inputs*** field, and you can choose which ones to use by dragging and dropping them to the ***Parameter*** field, as shown in the Figure above.

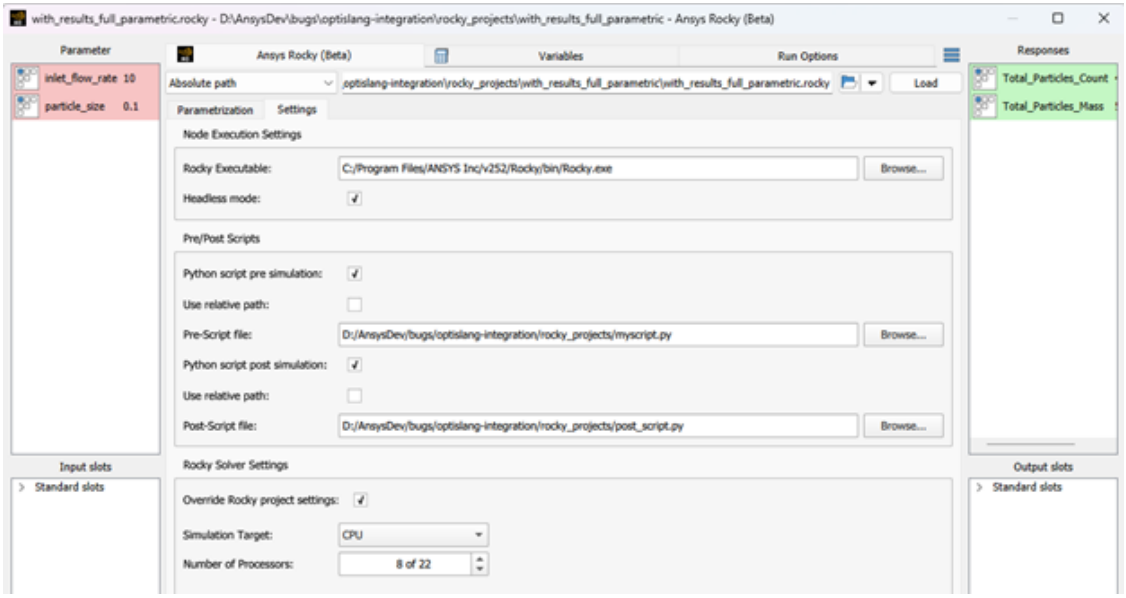
Similarly, the output parameters can be selected from the ***Output*** to the ***Response*** field.

In the "Settings" tab, you can set some options for node execution, namely:

- Changing the Rocky executable (by default, the plugin will use the Rocky executable in the ANSYS folder of the same version of optiSLang, if found).
- By default, Rocky will run in headless mode, but you can deactivate this option to have Rocky GUI launched for each design point during optiSLang executions.
- To define a script to be run inside Rocky before the simulation is executed ("Pre-Script file").
- To define a script to be run inside Rocky after the simulation is executed ("Post-Script file").
- Override Rocky Solver settings related to the "Simulation Target" (CPU/GPU).

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

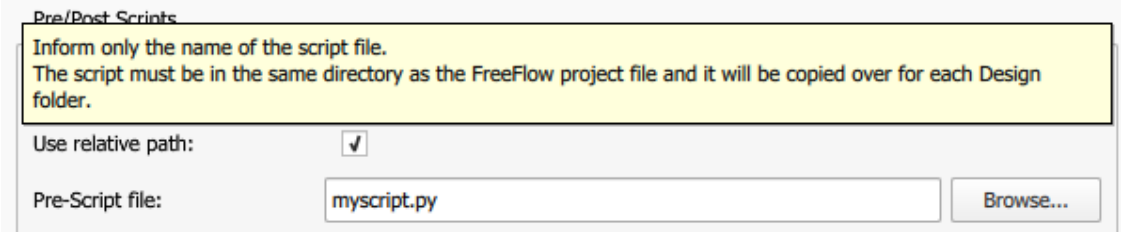
Figure 2.151: Ansys Rocky (Beta) Module Settings Tab



Note:

If the Pre/Post scripts are set to "use relative path," only the name of the file must be provided; it must be in the same directory as the .rocky file, and it will be copied to each Design folder (as is informed by a tooltip when hovering this setting in the UI).

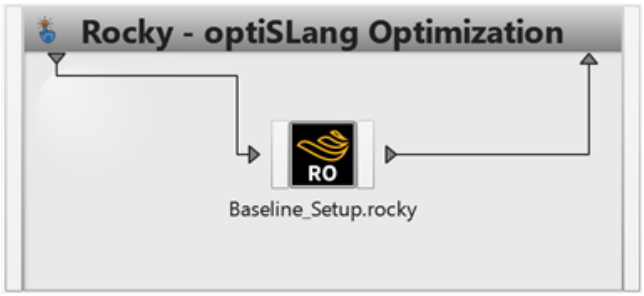
Figure 2.152: Ansys Rocky (Beta) Module PrePost Scripts



5. The Ansys Rocky (Beta) module will be available in the Schematic view, as shown in the Figure below. The parameters can be edited at any time by double-clicking the Rocky node.

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

Figure 2.153: Ansys Rocky (Beta) Module Schematic view



Tip:

For more information about using optiSLang, refer to the Ansys optiSLang User Documentation.

See Also:

- Understand Rocky and Ansys Integration (p. 143)

2.3.6. Ansys Rocky and Ansys Minerva

For more information about Rocky integration with Ansys Minerva, refer to the information on the Installation Guide. https://innovationspace.ansys.com/ais-rocky/.

See Also:

- Understand Rocky and Ansys Integration (p. 143)

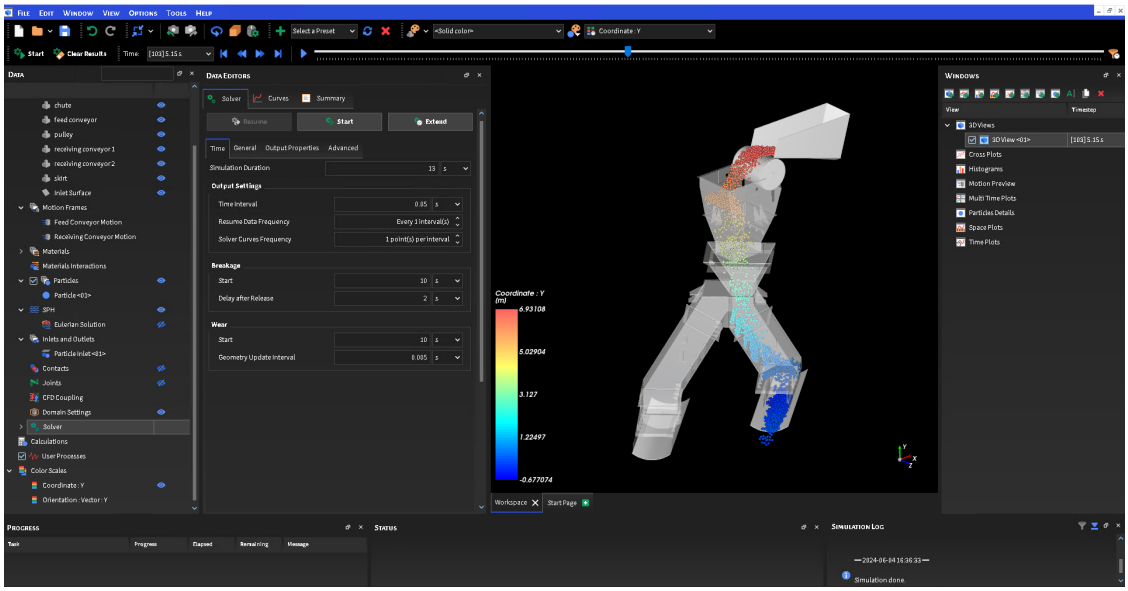
2.3.7. Ansys Rocky and Ansys EnSight

Ansys Rocky has its post-processing capabilities enhanced by coupling with **Ansys EnSight**. After running any case using the Rocky, with both DEM-SPH, you can open the .rocky file through the EnSight, and **post-process** using it all the **environment, lights, and materials processing capabilities.**

In the example below, the Transfer Chute case of our Rocky Tutorial 01 was normally solved in Ansys Rocky.

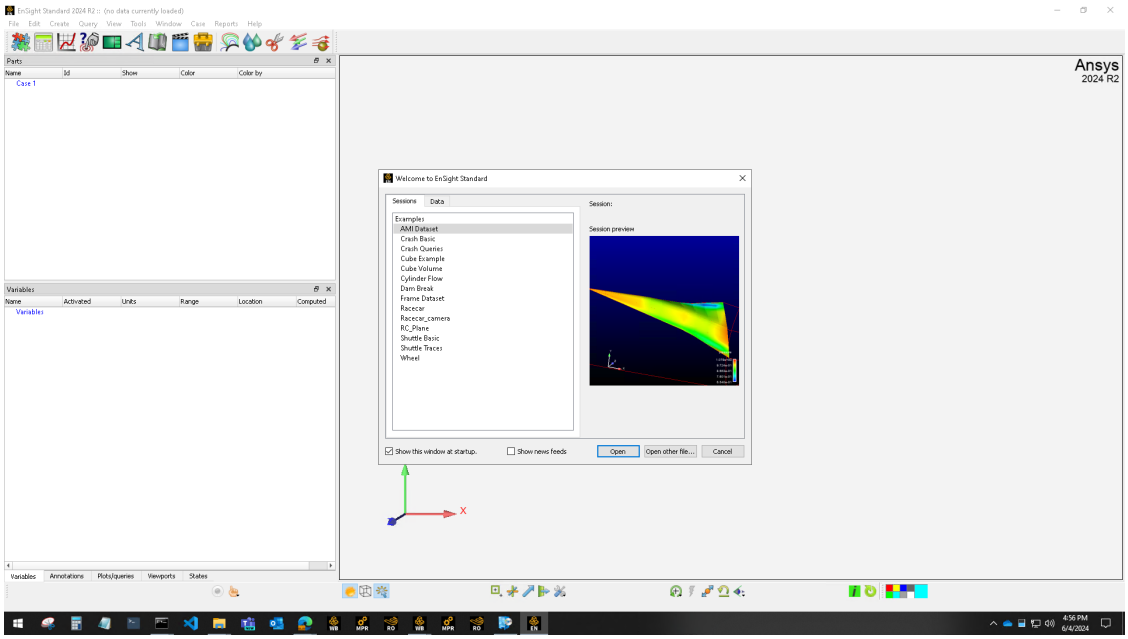
Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

Figure 2.154: Tutorial 01 inside Ansys Rocky



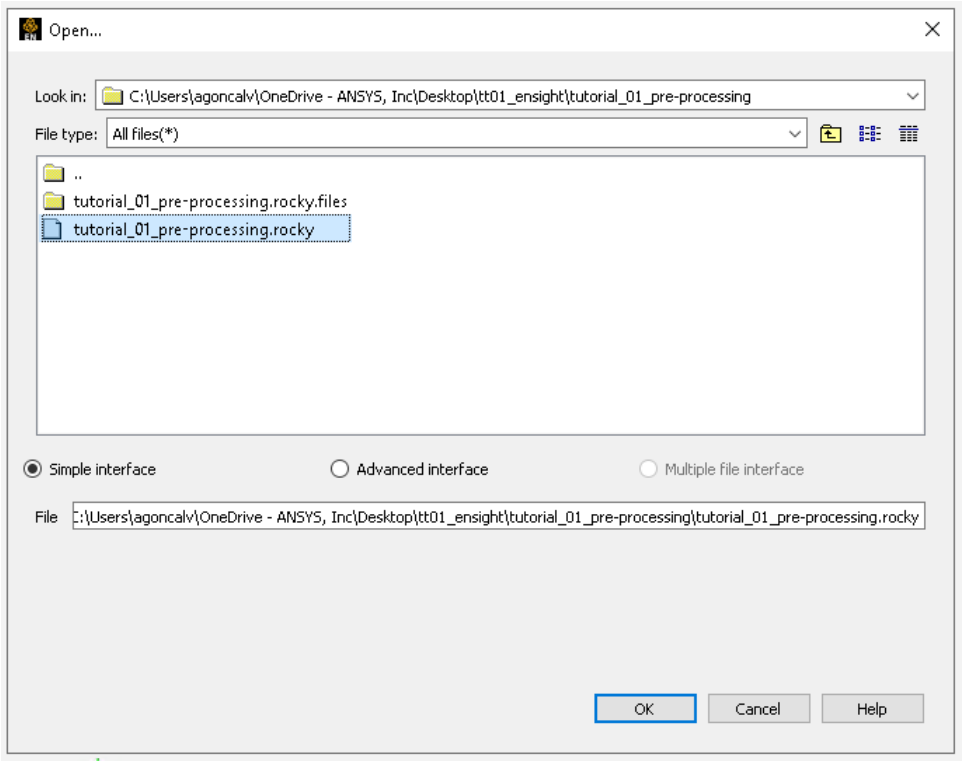
After fishing the simulation, the case was closed, and the Ansys EnSight was opened. The file that must be selected it is the **.rocky** file, as shown below:

Figure 2.155: Opening Ansys EnSight



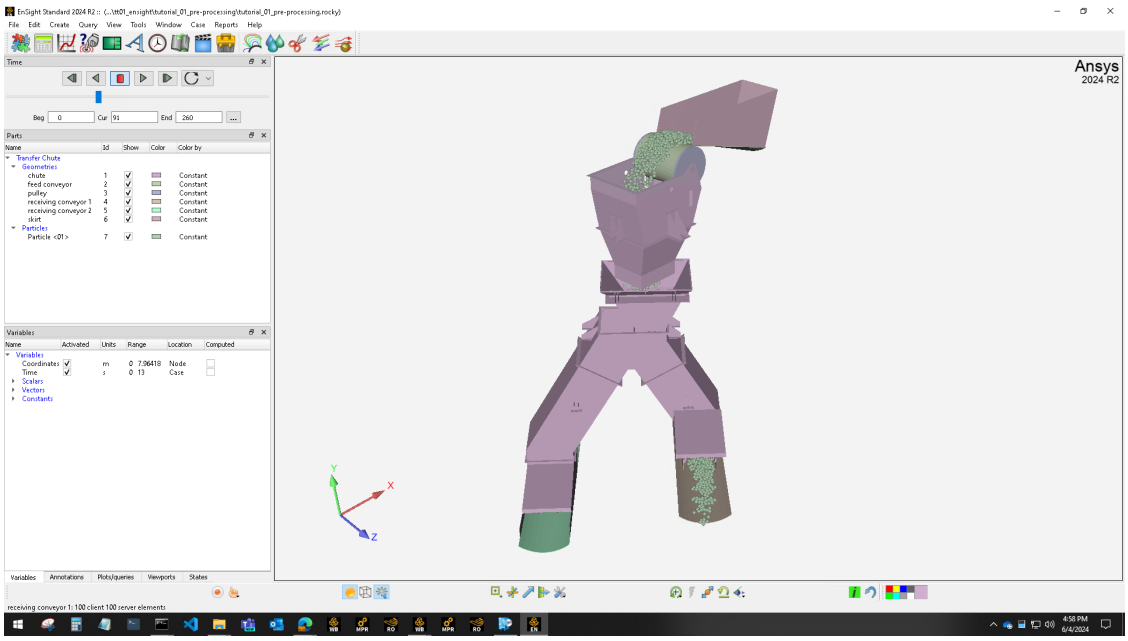
Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

Figure 2.156: Selecting the .rocky file



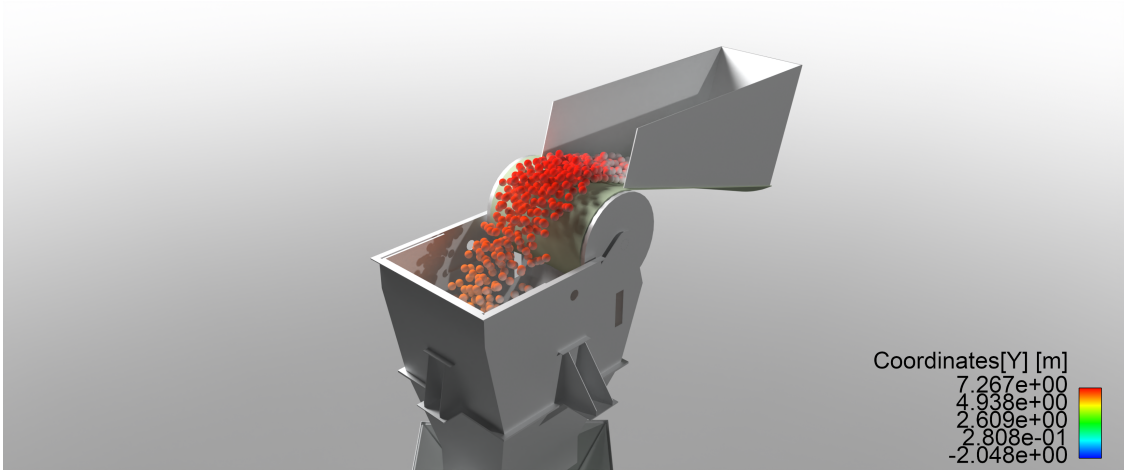
After that, the Transfer Chute case is normally opened by the EnSight, with all the results solved by the Rocky. The Ansys Rocky and EnSight coupling bring some **interesting benefits**, as the Ansys EnSight **rendering capabilities, including lighting, environment, and material presets**, as well as DEM-SPH representation enhancement.

Figure 2.157: Tutorial 01 inside Ansys EnSight



Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

Figure 2.158: Tutorial 01 render by Ansys EnSight



Note that all the Ansys Rocky cases that have the following characteristics **will not be able to be fully post-processed** by Ansys EnSight:

- **Geometry with Replication**
- **Properties that are not written in the disc (as the Pressure for the WCSPH, which are calculated in the outputs)**
- **Motion from FMU files (couplings with Motion, Adams, and other programs able to export FMU files)**
- **Deformable particles and Point Clouds**
- **1- and 2-way couplings with Fluent**

2.4. Ansys Rocky Modules

In Rocky, Modules (p. 1392) refer to separate pieces of code that when explicitly turned on (or enabled) prior to processing your simulation, add in discrete features or functionality within your project. This enables you to include-or even develop yourself-custom or specialized functionality into your Rocky projects without having to acquire a new version of the Rocky product itself.

There are several ways in which you can acquire Modules, and several ways Module functionality can affect how you set up your Rocky projects. See below sections for details.

In this version of Rocky, several features are provided via Modules, including Collision and Particle Statistics. Because these Modules are provided by default with your Rocky installation, these are sometimes referred to as embedded Modules (p. 1390).

Conversely, you may also have access to external Modules (p. 1391), which are custom Modules installed separately from the Rocky product.

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

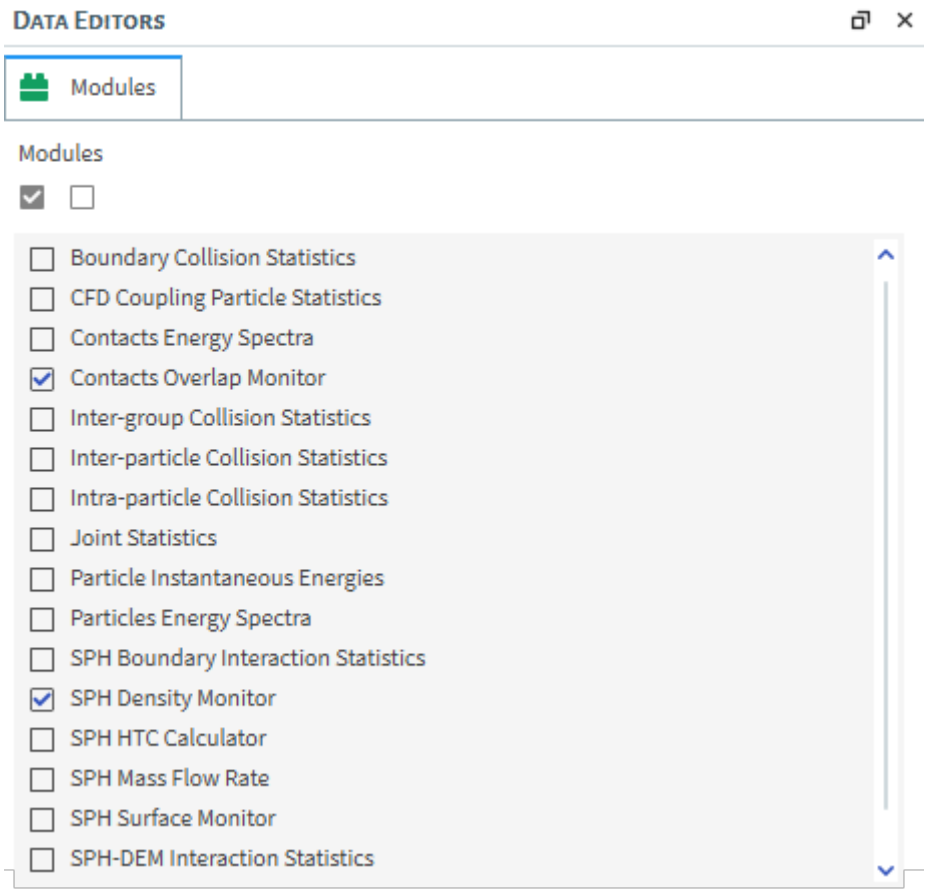
2.4.1. Internal Modules (Embedded)

The embedded Modules (p. 1390) included by default with Rocky (Figure 2.159: Modules available by default in Rocky (p. 222)) include the following:

- **Boundary Collision Statistics**. which enables the collection of boundary-related collision data, such as collision frequency, intensities, and impact velocities.
- **CFD Coupling Particle Statistics**, which enables the collection of fluid-particle interaction data.
- **Contact Energy Spectra**, which enables the collection of collision-based energy values during the simulation and the resulting data is categorized by the contact pair (particle group and/or geometry).
- **Contacts Overlap Monitor**, which checks each contact (p. 1390) pair (particle-particle or particle-boundary) for the amount that they overlapped-the percentage of which is determined by the size of the smallest particle in the contact pair-and raises a message in the Simulation Log panel if an overlap exceeds any of the three warning levels you define.
- **Inter-group Collision Statistics**, which enables the collection of energy dissipation data for each particle-particle and particle-boundary pair.
- **Inter-particle Collision Statistics**, which enables the collection of particle-related collision data between all particles in the simulation.
- **Intra-particle Collision Statistics**, which enables the particle-related collision data affecting the surfaces of a particular particle set.
- **Joint Statistics**, which enables the collection of joint-related statistical data.
- **Multibody Dynamics FMU Coupling**, which enables FMU files to be imported into Rocky without needing an external module for each compatible software.
- **Particle Instantaneous Energies**, which enables the calculation of the kinetic and potential energies of each individual particle in the simulation. These energies are useful when performing global or partial energy balances in a simulation.
- **Particles Energy Spectra**, which enables the collection of energy values during the simulation that are related only to particles, and the resulting data is classified by size and particle group.
- **SPH Boundary Interaction Statistics**, which enables SPH boundary-related data, generating properties of the forces that the fluid is exerting on the walls. Also, curves may be generated for the resultant forces, torques and power.
- **SPH Density Monitor**, which enables the monitoring of density values of the SPH elements during a simulation.
- **SPH HTC Calculator**, which calculates the heat transfer coefficient (HTC) through forced convection for each wall triangle.
- **SPH Mass Flow Rate**, which enables to measure the mass flow rate in the chosen surface.
- **SPH Surface Monitor**, which enables to track many SPH properties at a given simulation surface, while also allowing to define the output frequency of results.

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

Figure 2.159: Modules available by default in Rocky



Because these Modules are included with the Rocky product, they will be documented as usual in the User and Technical Manuals.

2.4.2. External Modules

You may have access to other external Modules (p. 1391) not included by default in Rocky. These might have been custom Modules created by you using the **Solver SDK** or modules created by Ansys Rocky team of specialists.

Tips: To learn more about making your own custom Modules, do one or more of the following:

- Access the Rocky **Solver SDK Manual**. (From the **Help** menu, point to **Manuals**, and then click **Solver SDK Manual**.)
- Complete Tutorial - Solver SDK in the *Rocky Tutorial Guide*

Rocky for Developers:https://developer.ansys.com/docs/rocky

External Modules are typically installed via ZIP file (see also Install an External Module (p. 223)). After installation, they will appear in Rocky under **Modules** in the **Data** panel.

Because these Modules are installed separately from the Rocky product, their documentation can be found in the Rocky Module Manual.

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

- Learn more about Rocky Simulation Entities that can be Affected by Modules (p. 223)
- Learn more About Modules Parameters (p. 431)

INSTALL AN EXTERNAL MODULE

Many more additional models and functionality are available as external Modules (p. 1391) that can be downloaded from the Ansys Customer Portal.

Note:

Even though this install procedure is similar on both Windows and Linux machines, be aware that Module ZIP files are specific to both the operating system and the version of Rocky for which the Module was created.

---

1. Locate or download the ZIP file for the external Module you want to install. **Tips**:

   Ensure that the ZIP file you download is appropriate for both the operating system and the Rocky version you are using.

2. Into your user folder's `..RockyModules` folder, extract the contents of the ZIP file. **For example**:

   - In Windows, this might be your `%USERPROFILE%DocumentsRockyModules` folder.
   - In Linux, this might be your `~/.Rocky/Modules` folder.

   Once extracted, the ZIP file will automatically create a build folder in the `Modules` folder and the Module contents will be installed there.

3. If the Rocky program is open, close it and open it again to refresh the Modules list. The external Module should be listed in the **Modules** entity. (From the **Data** panel, select **Modules** and then from the **Data Editors** panel, review the **Modules** list.)

See Also:

- Understand Rocky Modules (p. 220)
- Install an External Module (p. 223)
- Getting Started (p. 73)

2.4.3. Rocky Simulation Entities that can be Affected by Modules

Once you enable a Module (see also About Modules Parameters (p. 431)), there can be other places in the Rocky UI affected by that particular Module. How and what *simulation entities* (p. 1395) can be affected depends upon the Module and how it was built.

MODULE-RELATED EFFECTS TO THE ROCKY UI

Typically, enabling a Module can affect the Rocky UI in one or more of the following ways:

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

- **The Module overrides Rocky's default models or settings.** Models or settings that can be overwritten by a Module are considered to be exclusive (p. 1391). An exclusive model or setting allows a Module to disable its default options and replace them with the name of the Module.
- **The Module adds additional models or settings.** Models or settings that are added to the Rocky UI as a result of enabling a Module are considered to be non-exclusive (p. 1393). A non-exclusive model or setting allows a Module to add new options to its default set of options, but does not allow the Module to override any default options.

See the below sections for more detail about each of these two Module-related UI conditions. In addition, refer to Table 1 below to learn what areas of the Rocky UI might be affected by a Module.

EXCLUSIVE SETTINGS

Rocky UI settings that are considered to be exclusive (p. 1391) have the effect of disabling Rocky's default options for a particular model or setting, and then replacing them with the name of the Module. In these cases, you must use the model or setting defined in the enabled Module, and only one enabled Module can override any particular model or setting at one time. This means that if you happen to have more than one Module defining the same exclusive model or setting in different ways, you must enable only one of those Modules in your simulation setup.

NON-EXCLUSIVE SETTINGS

Rocky UI settings that are considered to be non-exclusive (p. 1393) have the effect of adding in additional models or settings for the Module, but do not override any default models or settings. Sometimes the additional models or settings appear on a separate **Modules** tab within the affected simulation entity, and sometimes new options are added to existing entities or lists.

No matter how these appear to you in the UI, if you have enabled a Module that includes additional models or settings, you must make use of those Module-related models or settings at least once in your simulation setup. This means that, for example, if you have two enabled Modules that define the same non-exclusive model for individual Particle sets in two different ways, you must define at least two different Particle sets to make use of the two different models that were added by their respective Modules.

SIMULATION ENTITIES THAT CAN BE AFFECTED BY MODULES

The table below lists the areas of the Rocky UI that are able to have their settings and options modified by a Module that is enabled.

Tip:

Refer to the Module's documentation for further details about Module-specific parameters. Specifically:

- For embedded Modules (p. 1390), this documentation can be found in the Rocky **User Manual** (the document you are reading now).
- For external Modules (p. 1391), this documentation (if provided) can be found by clicking the **Open this Module's Help File** icon (**?**) on the Module's main tab in the **Data Editors** panel.

Table 1: Simulation Entities that can be affected by Modules, and how they can be affected

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

Simulation Entity	Setting or Area Affected	How Affected	See Also
Physics | **Momentum** tab	Normal Force; Tangential Force; Adhesive Force; Impact Energy	Built-in model override (exclusive settings)	About Physics Parameters (p. 417)
Physics | **Thermal** tab	Heat Conduction Model; Thermal Integration Model	Built-in model override (exclusive settings)	About Physics Parameters (p. 417)
Individual imported **Geometries**	New **Modules** sub-tab	Additional settings added	
Individual imported **Geometries**	Wear | Wear Law	Additional models added (*non-exclusive* settings)	
Individual **Feed Conveyor** Geometries	New **Modules** sub-tab	Additional settings added	About Feed Conveyor Parameters
Individual **Receiving Conveyor** Geometries	New **Modules** sub-tab	Additional settings added	About Receiving Conveyor Parameters
Individual **Materials**	For each Material defined under **Materials**, a new group box labeled with the Module's name	Additional settings added	About Modifying Material Compositions (p. 591)
Materials Interactions	For each pair of materials interactions, a new group box labeled with the Module's name	Additional settings added	About Modifying Materials Interactions and Adhesion Values (p. 597)
Individual **Particle** set	Composition | Joint Model; Breakage | Model; Breakage| Fragment distribution | Distribution model	Additional models added (*non-exclusive* settings)	About Adding and Editing Particle Sets (p. 602)
Individual **Particle** set	New **Modules** sub-tab	Additional settings added	About Adding and Editing Particle Sets (p. 602)
Inputs	New **Modules** sub-tab	Additional settings added	About Adding and Editing Particle Inputs (p. 708)

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

Simulation Entity	Setting or Area Affected	How Affected	See Also
CFD Coupling | **1-Way Fluent** tab; **CFD Coupling** | **2-Way Fluent** tab; **CFD Coupling** | **1-Way Constant** tab	Interactions | Drag Law; Interactions | Lift Law; Interactions | Torque Law; Interactions | Virtual Mass Law; Interactions | Convective Heat Transfer Law	Additional models added (*non-exclusive* settings)	About Using the 1-Way Fluent Method (p. 775); About Using the 2-Way Fluent Method (p. 795); About Using the 1-Way Constant Method (p. 763)

See Also:

- About Modules Parameters (p. 431)
- Install an External Module (p. 223)
- Getting Started (p. 73)

2.5. Ansys Rocky Preferences

There are several parameters you can set in Rocky that will affect your overall simulation. These include various preferences and unit types, as well as variables and scripts. These items can be determined before you set up your simulation, or can be modified at any point during the setup process.

What would you like to learn about?

- Setting Your Global Preferences (p. 226)
- Setting Your Unit System (p. 238)
- Defining Your Variables (p. 243)
- Creating and Using PrePost Scripts (p. 296)

See Also:

- Getting Started (p. 73)

2.5.1. Setting Your Global Preferences

Global preferences for Rocky include settings like memory usage, date and time format, video, plot and view options. What you choose to set in preferences will affect the rest of the Rocky user interface.

What would you like to do?

- Learn more About Setting Global Preferences (p. 227)
- Set Global Preferences (p. 238)

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

ABOUT SETTING GLOBAL PREFERENCES

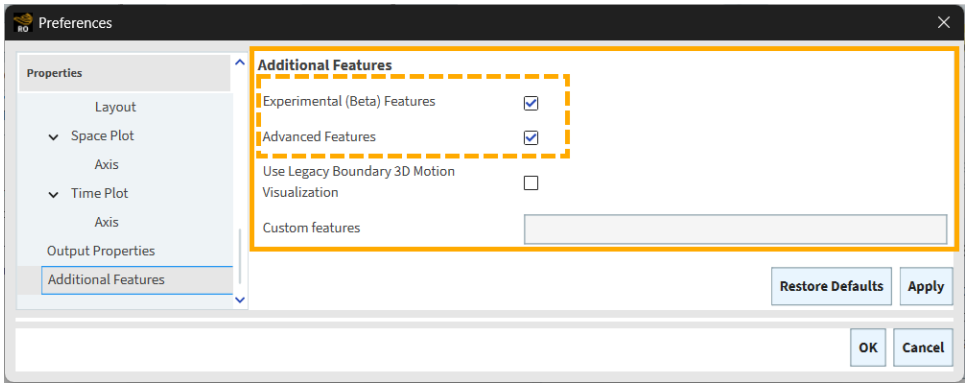
When you set your global preferences, you determine what global (default) options you want to set for memory usage, date format, format, drag and drop, Internet proxy, video encoding and quality, and display for Rocky windows, including 3D Views, plots, and histograms. You can also view (but not edit) what keyboard shortcuts perform which commands.

The settings that you choose in the Preferences dialog will persist throughout your entire Rocky experience. Some settings, like memory usage and Internet proxy settings, can only be changed from the Preferences dialog. Other settings, like window displays, affect only what default settings appear in the various panels within the Rocky UI but can be overridden from the panel whenever you want.

Important: Changes you make within the Preferences dialog may not immediately affect in-process features or functionality. To ensure that all changes you make within this dialog are applied immediately, click **OK** on the Preferences dialog, and then close and reopen any Rocky windows, panels, or projects affected by the Preferences changes you made.

ADDITIONAL FEATURES

Figure 2.160: Additional Features



Experimental (Beta) Features

The **Experimental (Beta) Features** checkbox enable you to use Rocky beta features, these features are described in this document with the reference that they are still beta and will only appear in the UI if this check box is activated.

Important:

Note that beta features have not been fully tested and validated. Ansys, Inc. makes no commitment to resolve defects reported against these prototype features. However, your feedback will help us improve the overall quality of the product. We will not guarantee that the projects using this beta feature will run successfully when the feature is finally released so you may, therefore, need to modify the projects.

Use Legacy Boundary 3D Motion Visualization

Enable this option to revert to the previous visualization method. This legacy mode is slower and should only be used for compatibility purposes, as the new default visualization provides faster 3D post-processing for large geometries.

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

Advanced Features

The **Advanced Features** checkbox enable you to use Rocky Advanced features, these features are described in this document with the reference that they are advanced and will only appear in the UI if this check box is activated.

PREFERENCES FOR NON-WINDOWS PROPERTIES

See the images and table below to help you understand what to set for your global preferences on properties not including those on the **Windows** panel.

PREFERENCES FOR WINDOWS PROPERTIES

The Properties settings listed under **Windows** on the Preferences dialog affect only the configuration used to create windows (p. 1396) of the selected type. Any of these default settings can be overridden from the **Window Editors** panel whenever you want.

See the images and tables below to help you understand what global preferences to set for Windows panel-related items.

Also, see additional information on these settings in the following topics:

- About Using the Window Editors Panel to Change All 3D Views in the Project (p. 1048)
- About Using the Window Editors Panel to Change the Selected 3D View (p. 1058)
- About Changing the Appearance of a Graph (Plot or Histogram) (p. 1126)

Figure 2.161: Preferences, 3d View

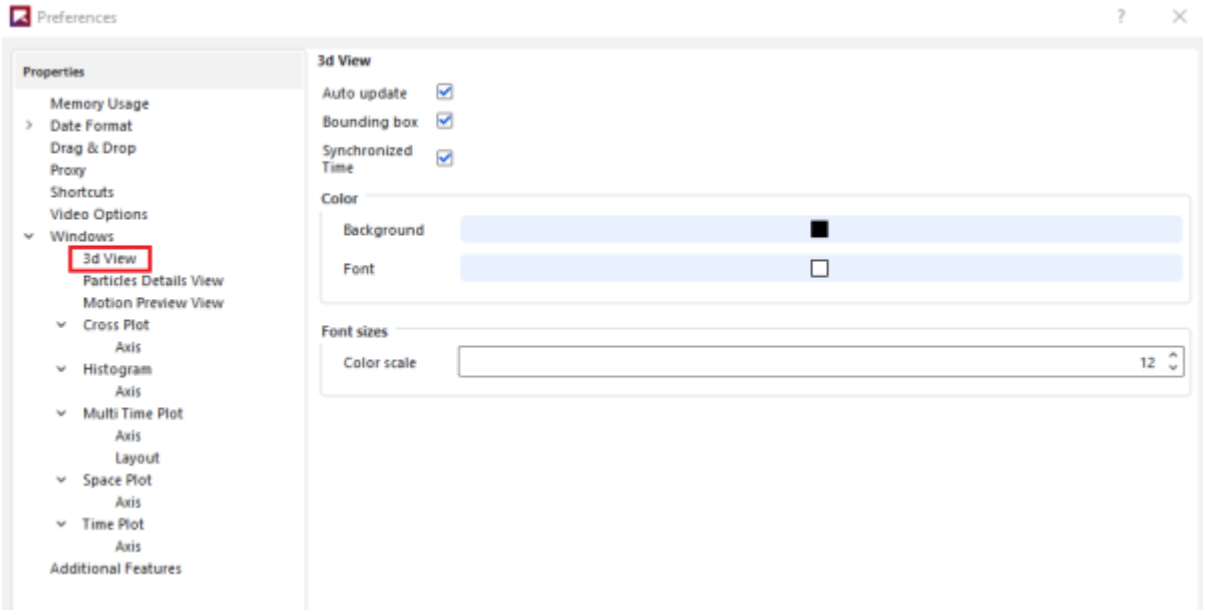


Table 2: Preferences options for 3d View Windows

Property	Description	Range
Auto update	When enabled, any change made to the display settings of the selected 3D View or to the items affecting the data or	Turns on or off

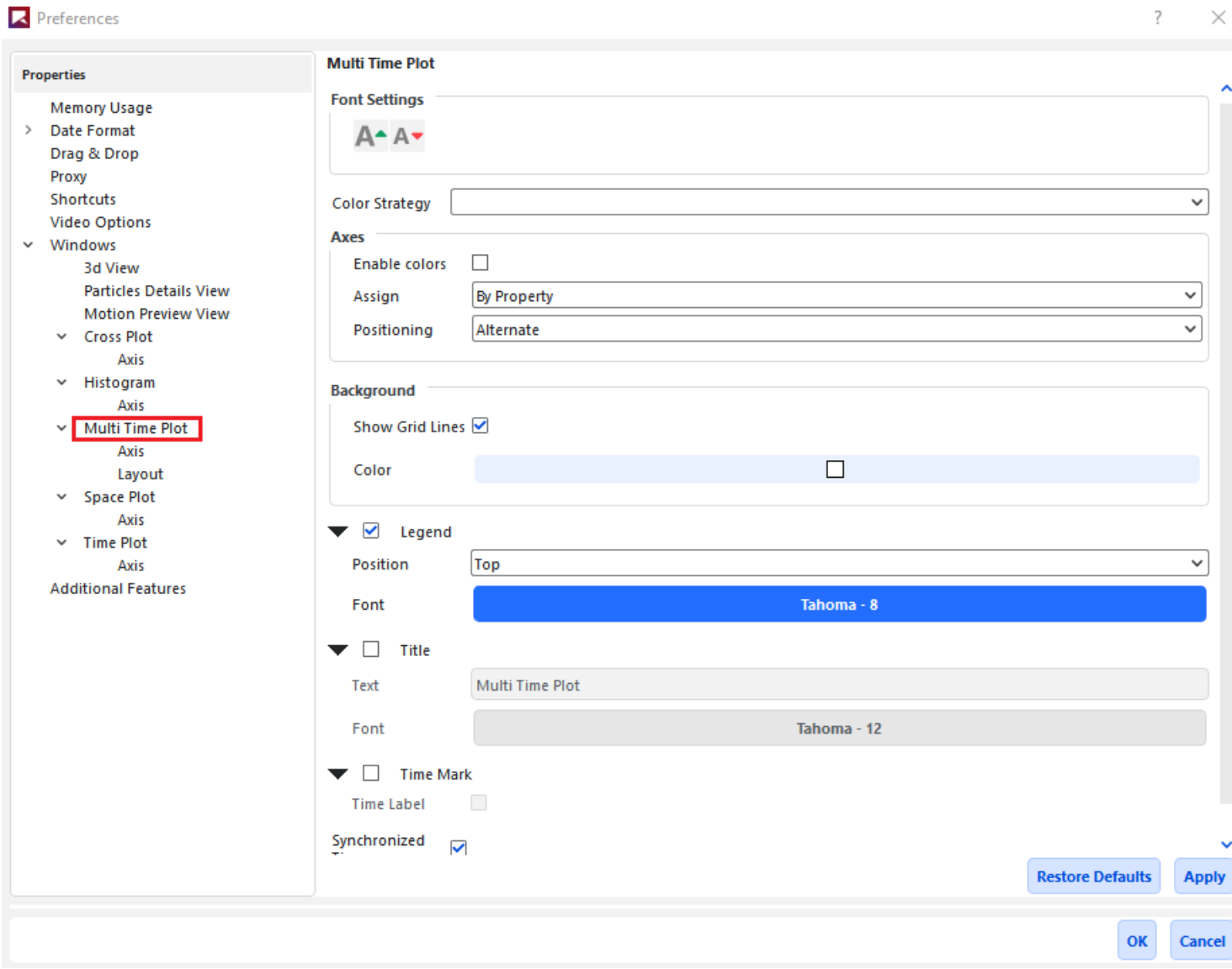
Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

Property	Description	Range
	settings calculated and displayed within the 3D View, including the output time or Properties displayed, will be updated in the 3D View automatically. Because automatically updating the view with each separate change can be computationally intensive, you can clear this option to have all calculated items and more computationally intensive display options remain unchanged in the 3D View until you enable **Auto update** again. **Tip:** You will know when the calculations in a 3D View window are not being updated when you see a thick red border around the window.	
Bounding box	When enabled, displays measurements illustrating the simulation entity (p. 1395) in the selected 3D View.	Turns on or off
Synchronized Time	When enabled and the window (or any other window with this checkbox enabled) is selected, the details shown in this (and any other Synchronized Time) window will be updated when the current output time is changed on the Time toolbar. (See also About the Time Toolbar (p. 1279).) When cleared and the window is selected, only this window will be updated when the current output time is changed on the Time toolbar. **Tip:** To keep the output time synchronized between multiple **3D View** or other windows, ensure each window has this option enabled.	Turns on or off
Color Background	Enables you to change the color that appears behind the geometries and particles in the 3D View.	Options limited by the choices in the **Select Color** dialog
Color Font	Enables you to change the color of the labels, borders, and axes lines displayed in the 3D View. **Note:** These color options can be overridden for text overlays by using the **Color** option on the **Overlays** tab. (See Table 3 below.)	Options limited by the choices in the **Select Color** dialog

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

Property	Description	Range
Font sizes Color scale	Enables you to change the size of the text used on color scales for Properties, which, unlike the other lines or labels displayed on the 3D View, is not affected by the window size or the level of zoom you have applied.	Whole values from 1 to 100

Figure 2.162: Preferences, Multi Time Plot (similar for all plots)



Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

Figure 2.163: Preferences, Histogram

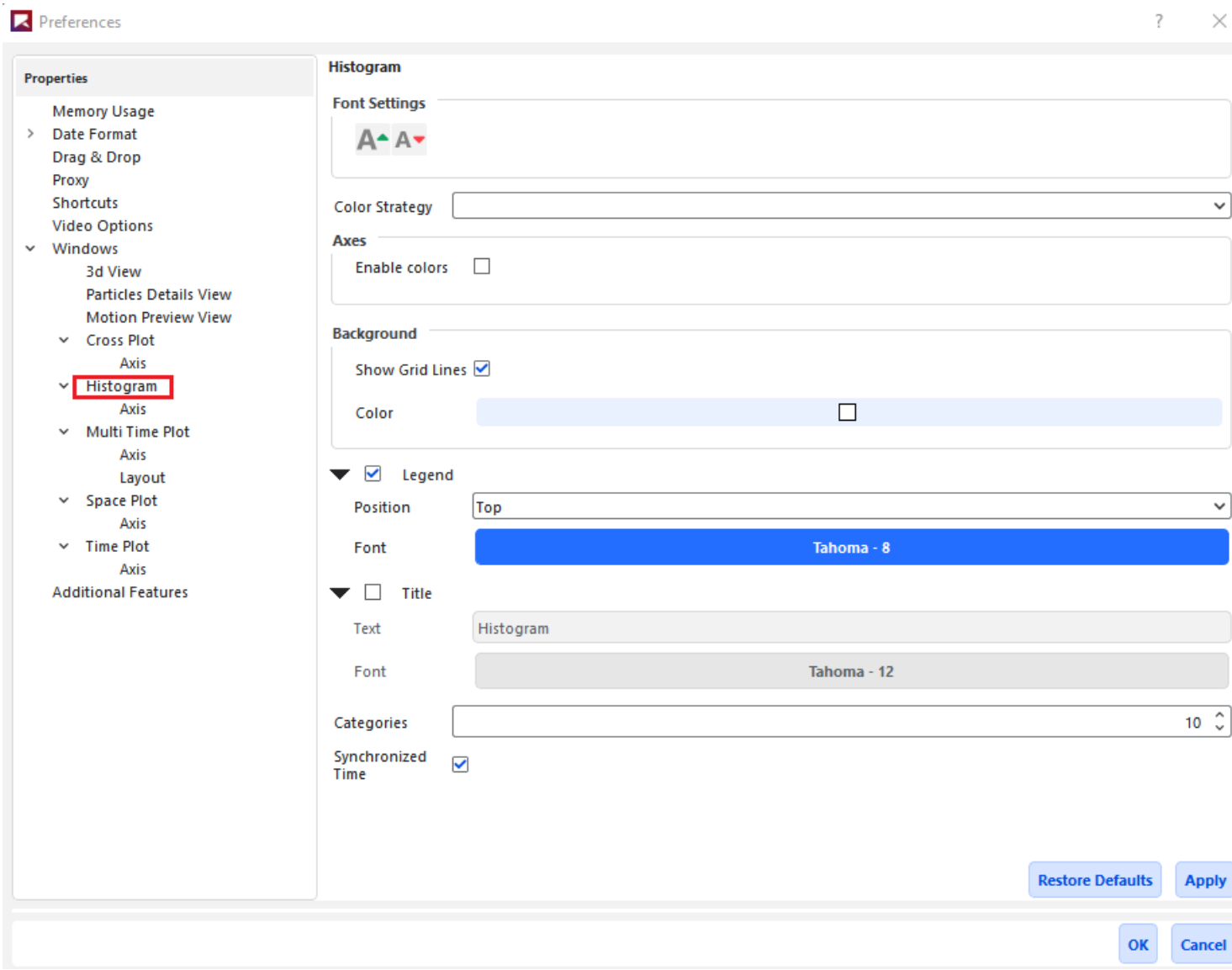


Table 3: Preferences options for Graph (Plot and Histogram) Windows

Setting	Description	Range
Font Settings	The following two buttons in this section apply to all axes, legend, and title text shown in the graph: • **Increase font size**: Makes text one font-size bigger than current display. • **Reduce font size**: Makes text one-font-size smaller than current display. Notes:	Increase font size; Reduce font size

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

Setting	Description	Range
	• These buttons do not affect the text for Annotations. (See the **Font** setting under **Annotations** | **Text** for more information. • You can also change the size of the legend text, the title text, and the axis title and value label text by using those attributes' own **Font** settings. (See further down this table for more information.)	
Color Strategy	Changes the coloring criteria of the curves, which affects the legend text and plot line of the properties (Property or Curves) being graphed. This option can be used to group curves with a similar representation by color based upon the following options: • **Entity Based**: Indicates the geometry, Particle set, or user process upon which the data is based. • **Property Based**: Indicates the Property or Curve upon which the data is based. (See also About Creating and Saving an Animation (p. 1166).) • **Unique**: Indicates all available color strategy information including entity, and property based.	Entity Based; Property Based; Unique
Axes Enable colors	Enabling this checkbox colors the Y axes according to the **Color Strategy** selected. Clearing this checkbox keeps all axes colored black.	Turns on or off
Axes Assign	For plots with multiple Properties or Curves, this enables the Y axes to be shared according to the following options: • **By Quantity**: Any properties with a similar type of measurement- no matter the unit assigned-(for example, length, volume, or pressure) will share the same axis. • **By Property**: Each unique property will have its own axis on the plot.	By Quantity; By Property
Axes Positioning	For plots only, this sets on which side of the graph the axes will be displayed. Specifically: • **Alternate** switches locations-left or right of the plot- with each new axis added to the plot. • **Left Axes First** places the first four axes to the left of the plot, then switches to the right. • **Right Axes First** places the first four axes to the right of the plot, then switches to the left.	Alternate; Left Axes First; Right Axes First

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

Setting	Description	Range
Background Show Grid Lines	Selecting this checkbox enables black grid lines to appear behind your graph.	Turns on or off
Background Color	Enables you to choose the color that appears behind your graph.	Options limited by the **Select Color** dialog
Legend	Enables the legend information, the content and color of which are determined by the **Color Strategy** setting, to be displayed on the graph window.	Turns on or off
Legend Position	When **Legend** is enabled, determines where the Legend information appears on your graph.	Bottom; Left; Right; Top
Legend Font	Enables you to change the type, style, size, and effects of the font used for displaying the Legend text. **Note:** The size you choose will still be affected by the **Font Settings** buttons.	Options limited by the **Select Font** dialog
Time Mark	For Time and Multi Time Plots only, enables a dotted, vertical line to be displayed at the location of the current output time.	Turns on or off
Time Label	When **Time Mark** is selected, enables you to add a label to the mark displaying the current output time value. **Note:** The value displayed reflects the units used for the Time (X) axis.	Turns on or off
Title	Enables you to add a separate title to the top of the graph.	Turns on or off
Title Text	When **Title** is selected, determines what text will be displayed in the title.	No limit
Title Font	Enables you to change the type, style, size, and effects of the font used for displaying the Title text. **Note:** The size you choose will still be affected by the **Font Settings** buttons.	Options limited by the **Select Font** dialog
Categories (same as **Number**	For histograms only, enables you to choose the number of bins or bars the data is aggregated into.	Whole numbers greater than 1

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

Setting	Description	Range
of Bins elsewhere)		
Synchronized Time	When enabled and the plot window (or any other window with this checkbox enabled) is selected, the details shown in this (and any other Synchronized Time) window will be updated when the current output time is changed on the Time toolbar. (See also About the Time Toolbar (p. 1279).) When cleared and the window is selected, only this window will be updated when the current output time is changed on the Time toolbar. **Tip:** To keep the output time synchronized between multiple plot or other display windows, ensure each window has this option enabled.	Turns on or off

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

Figure 2.164: Preferences, Cross Plot Axis (similar for all graphs)

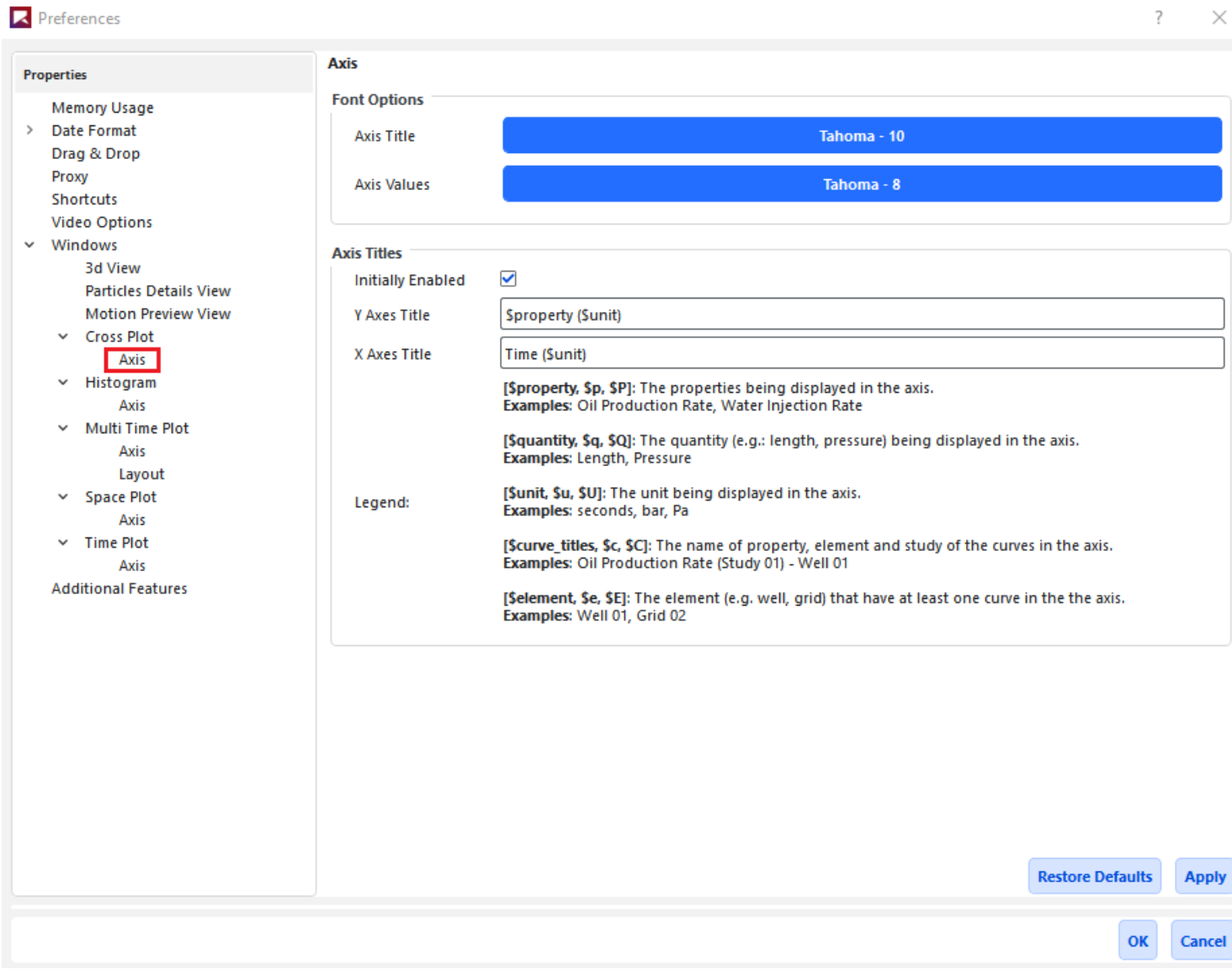


Table 4: Preferences options for Axes on Graph (Plot and Histogram) Windows

Setting	Description	Range
Font Options		
Axis Title	Enables you to change the type, style, size, and effects of the font used for displaying the title **Text** for the selected axis. **Note:** The size you choose will still be affected by the **Font Settings** buttons located on the first tab of the Window Editors panel.	Options limited by the **Select Font** dialog

Setting	Description	Range
Axis Values	Enables you to change the type, style, size, and effects of the font used for displaying the value marks for the selected axis. **Note:** The size you choose will still be affected by the **Font Settings** buttons located on the first tab of the Window Editors panel.	Options limited by the **Select Font** dialog
Axis Titles		
Initially Enabled	When selected, default axis titles will be added to the plot. When cleared, no axis titles will appear by default; you must manually turn on **Axis Title** from the Window Editors panel when you want an axis title to show.	Turns on or off
Y Axes Title	Enables you to specify the default format for titles appearing on the Y axis. Any Keywords you specify will be replaced with the corresponding data in the **Axes Title** field, and will be updated when the data supporting the Keyword changes. **Tip:** Sample Keywords are provided in the **Legend** section.	No limit
X Axes Title	Enables you to specify the default format for titles appearing on the X axis. Any Keywords you specify will be replaced with the corresponding data in the **Axes Title** field, and will be updated when the data supporting the Keyword changes. **Tip:** Sample Keywords are provided in the **Legend** section.	No limit
Legend	Displays sample Keywords you can use when defining **Axes Title** information.	Display only

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

Figure 2.165: Preferences, Multi Time Plot Layout

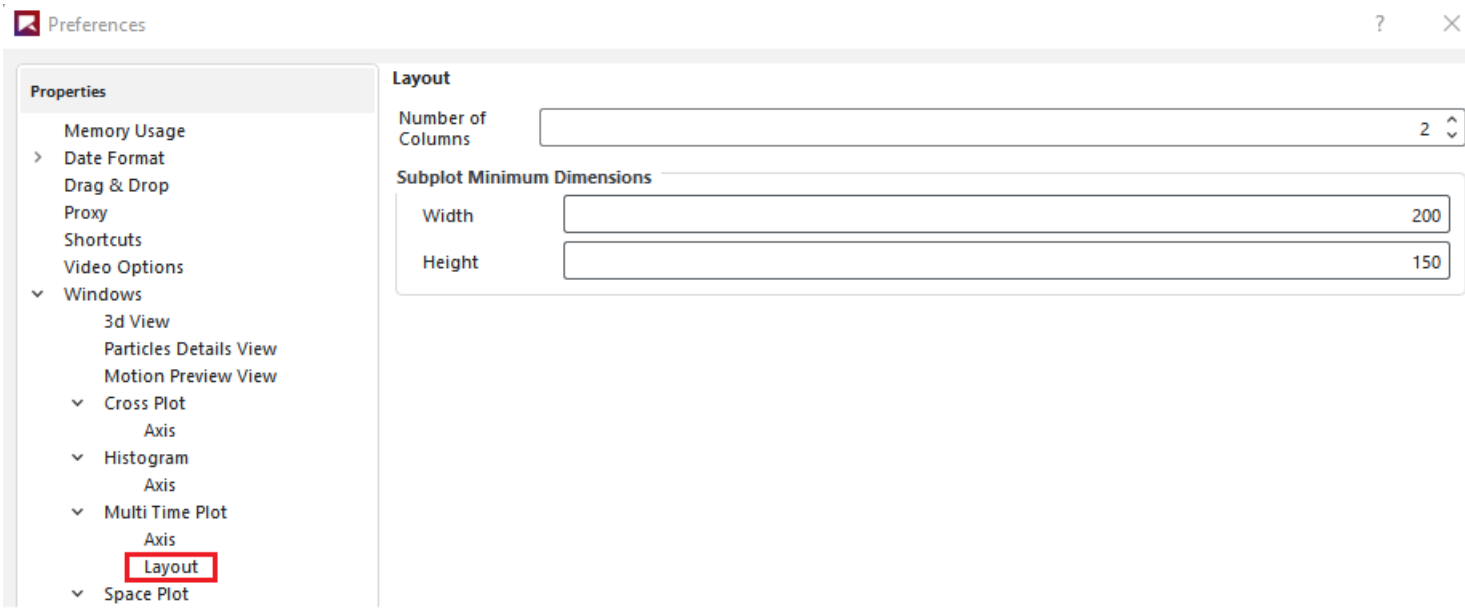


Table 5: Preferences options for Layout on Multi Time Plot Windows

Setting	Description	Range
Number of Columns	Enables you to set how many columns into which you want the sub-plots of your selected Multi Time Plot arranged.	Whole numbers greater than or equal to one
Subplot Minimum Dimensions Width	Enables you to define the minimum pixel width you want Rocky to maintain for each individual sub-plot in the selected Multi Time Plot. When making the window size smaller, the sub-plots will continue to be displayed in full until this setting is reached, at which point scroll bars will be added.	Whole values greater than or equal to 300
Subplot Minimum Dimensions Height	Enables you to define the minimum pixel height you want Rocky to maintain for each individual sub-plot in the selected Multi Time Plot. When making the window size smaller, the sub-plots will continue to be displayed in full until this setting is reached, at which point scroll bars will be added.	Whole values greater than or equal to 150

What do you want to do?

- Set Global Preferences (p. 238)

See Also:

- Set Your Rocky Preferences (p. 226)
- About Using the Window Editors panel to Change a 3D View (p. 1058)
- About Changing the Appearance of a Graph (Plot or Histogram) (p. 1126)

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

- I cannot find Particle Size Scale Factor (p. 1330)

SET GLOBAL PREFERENCES

1. From the **Options** menu, click **Preferences**.
2. From the **Preferences** dialog, choose the category you want under **Properties**, and then make the choices you want on the right. Note: Shortcuts are not currently editable.
3. Click **Apply** to save your changes.
4. Repeat steps 2-3 until you have set all the options you want, and then click **OK** to close the dialog. Tip: To ensure that all changes you made within this dialog are applied immediately, close and reopen any Rocky windows, panels, or projects affected by the Preferences changes you made.

See Also:

- About Setting Global Preferences (p. 227)
- Set Your Rocky Preferences (p. 226)

SETTING YOUR UNIT SYSTEM

Before you begin using Rocky, it is important that you understand the units used in the user interface and set them the way you want.

What would you like to do?

- Learn About Setting Your Unit System (p. 238)
- Select a Unit System to Use Throughout Rocky (p. 241)
- Create a New Unit System (p. 241)
- Edit a Custom Unit System (p. 242)
- Remove a Custom Unit System (p. 242)

See Also:

- Set Your Rocky Preferences (p. 226)

ABOUT SETTING YOUR UNIT SYSTEM

Currently, there are two unit systems included in Rocky: the International System and the English. You can choose to use one of these two included systems, or you can create your own custom unit system.

Whatever system you choose will provide the default unit settings for all parameters used within Rocky. However, each individual parameter unit can still be changed manually to another unit type at any time.

Use the images and tables below to help you understand your unit system options.

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

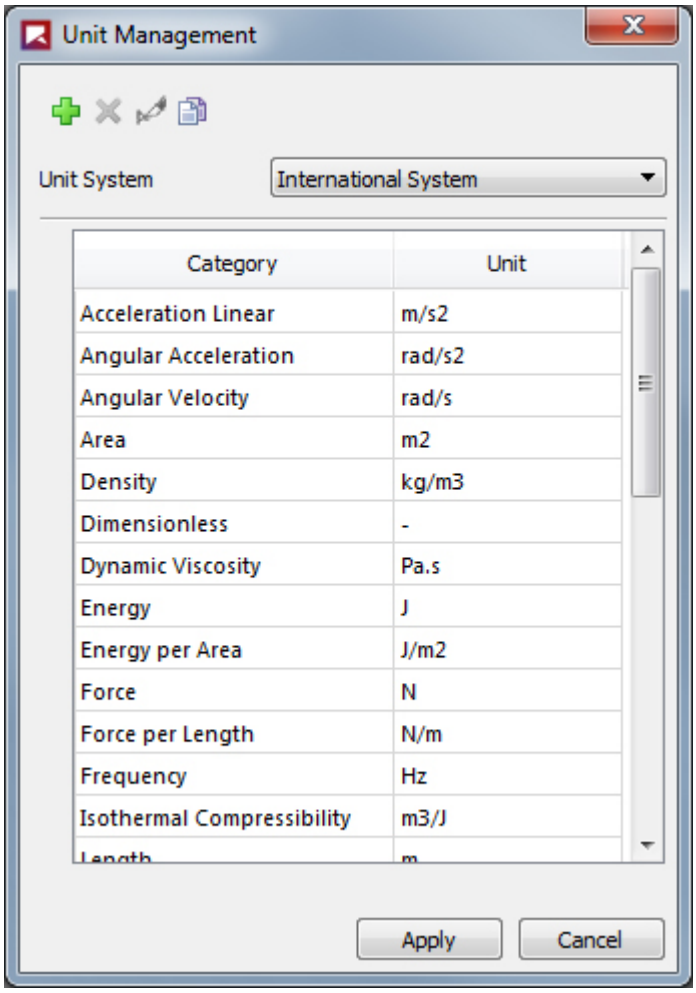


Figure 1 : Unit Management dialog

Table 1: Unit Management dialog options

Setting	Description	Range
Unit System	Sets the unit system used by default throughout Rocky. Choices include: • **International System**: Units based upon the International System of Units (SI). • **English Units**: System based on the customary units of measurement used in the United States. If you have created custom unit systems, those will display in this list also.	Lists all default and custom unit systems for Rocky.
Category	Lists the descriptive name of the item being measured.	Automatic
Unit	Provides the unit symbol that will be used by default for the parameters in Rocky. These are editable when you choose to create a custom system.	Automatic

Table 2: Default Units Used in Rocky

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

Item	International Unit	International Unit Symbol	English Unit	English Unit Symbol
Acceleration Linear	meter per second squared	m/s^2	foot per second squared	ft/s^2
Angular Acceleration	radian per second squared	rad/s^2	radian per second squared	rad/s^2
Angular Velocity	radian per second	rad/s	radian per second	rad/s
Area	square meter	m^2	square foot	ft^2
Density	kilogram per cubic meter	kg/m^3	pound mass per cubic foot	lb_m/ft^3
Dimensionless	-	-	-	-
Dynamic Viscosity	Pascal second	Pa·s	centipoise	cP
Energy	joule	J	foot pound force	$ft\cdot lb_f$
Energy per Area	joule per meter squared	J/m^2	pound force per foot	lb_f/ft
Force	Newton	N	pound force	lb_f
Force per Length	Newton per meter	N/m	pound force per foot	lb_f/ft
Frequency	hertz	Hz	hertz	Hz
Isothermal Compressibility	cubic meter per joule	m^3/J	cubic meter per joule	m^3/J
Kinematic Viscosity	square meter per second	m^2/s	centistoke	cSt
Length	meter	m	foot	ft
Mass	kilogram	kg	pound mass	lb_m
Mass Flow Rate (formerly "Tonnage")	metric tons per hour	t/h	pound mass per hour	lb_m/h
Mass per Energy	kilogram per joule	kg/J	pound mass per foot pound force	lb_m / $ft\cdot lb_f$
Moment of Force (formerly "Torque")	Newton meter	N·m	foot pound force	$ft\cdot lb_f$
Moment of Inertia	kilogram meter squared	$kg\cdot m^2$	square foot pound mass	$lb_m\cdot ft^2$
Percentage	Percentage	%	Percentage	%
Plane Angle	angular degree	dega	angular degree	dega
Poisson Ratio	-	-	-	-
Power	watts	W	horsepower	hp

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

Item	International Unit	International Unit Symbol	English Unit	English Unit Symbol
Power per Mass	watts per kilogram	W/kg	watts per kilogram	W/kg
Pressure	Pascal	Pa	pound-force per square inch	psi
Specific Energy	joules per kilogram	J/kg	foot pound force per pound mass	ft· lb_f /lb_m
Thermodynamic Temperature	Kelvin	K	degrees Fahrenheit	degF
Time	second	s	second	s
Velocity	meter per second	m/s	feet per second	ft/s
Volume	cubic meter	m^3	cubic feet	ft^3
Yield Stress	force per unit area	N/m^2	pound force per square foot	lb_f /ft^2

What would you like to do?

- Select a Unit System to Use Throughout Rocky (p. 241)
- Create a New Unit System (p. 241)
- Edit a Custom Unit System (p. 242)
- Remove a Custom Unit System (p. 242)

See Also:

- Set Your Rocky Preferences (p. 226)

SELECT A UNIT SYSTEM TO USE THROUGHOUT ROCKY

1. From the **Options** menu, click **Unit System Manager**.
2. From the **Unit Management** dialog, choose the option you want from the **Unit System** list.
3. Click **Apply** to save your changes and close the dialog.

See Also:

- About Setting Your Unit System (p. 238)

CREATE A NEW UNIT SYSTEM

1. From the **Options** menu, click **Unit System Manager**.
2. From the **Unit Management** dialog, do one of the following:

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

- To create a new unit system based upon an existing one, choose the option you want to copy from the **Unit Systems** list, and then click the **Copy the current Unit System** button. A copy of the existing system is created.
- To create a new blank system, press the **Create a new Unit System** button. A blank system is created.

3. Click the **Rename the current Unit System** button and then in the **Enter the new name** box, type the name you want and then click **OK**.
4. Edit the system list as you want.
5. Click **Apply** to save your changes and close the dialog.

See Also:

- About Setting Your Unit System (p. 238)

EDIT A CUSTOM UNIT SYSTEM

1. From the **Options** menu, click **Unit System Manager**.
2. From the **Unit Management** dialog, choose the custom unit system you want to edit from the **Unit System** list. Note: You can edit only the custom unit systems that you have added. If you want to edit the International or English system units included in Rocky, you must first make a copy of those systems and then edit the copy.
3. Do one or more of the following:
   - To change the units used, choose the option you want from the **Unit** list.
   - To add a new category, click the **Add** button, and then from the **Add Category** dialog, choose the options you want in the **Category** and **Unit** lists, and then click **OK**.
   - To remove a category, select the category you want to remove and click the **Remove** button.
4. Click **Apply** to save your changes and close the dialog.

See Also:

- About Setting Your Unit System (p. 238)

REMOVE A CUSTOM UNIT SYSTEM

1. From the **Options** menu, click **Unit System Manager**.
2. From the **Unit Management** dialog, choose the custom unit system you want to remove from the **Unit System** list, and then click the **Remove the Current Unit System** button. Note: You can only remove custom unit systems that you have created. The International and English system units included in Rocky are not removable.
3. Click **Apply** to save your changes and close the dialog.

See Also:

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

- About Setting Your Unit System (p. 238)

2.5.2. Defining Your Variables

The Expressions/Variables tool in Rocky enables you to define unique placeholders for both Input variables (values used within the Rocky set up parameters) and Output variables (values that are exported out of Rocky into programs like Ansys Workbench for tasks like structural analysis). The variables you define can then be used in place of actual numerical values within the parameter text fields. This is useful in cases where you have common values that are linked (all belts are to be the same width, for example), and/or the values you enter might be changing later in the setup (you want to experiment with three different belt speeds, for example).

What would you like to learn more about?

- Definng and Using Input Variables (p. 243)
- Defining Output Variables (p. 250)
- Rocky and Ansys Workbench Integration (p. 144)

See Also:

- Set Your Rocky Preferences (p. 226)

DEFINING AND USING INPUT VARIABLES

Using Input variables is also a good way to make complicated project setups simpler and easier to verify then by manually entering the same information into many different parameter field locations.

What would you like to do?

- Learn About Defining and Using Input Variables (p. 243)
- Create a New Input Variable from within a Parameter Field (p. 247)
- Edit the Value of an Existing Input Variable (p. 248)
- Edit the Name of an Existing Input Variable (p. 248)
- Change the Units for an Input Variable-or Functions Using Input Variables-in Use by a Field (p. 249)
- Remove a Single Input Variable (p. 249)
- Remove All Input Variables (p. 249)

See Also:

- About Defining Output Variables (p. 250)
- Set Your Rocky Preferences (p. 226)

ABOUT DEFINING AND USING INPUT VARIABLES

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

For Input variables, you can enter into the parameter text fields a variable name on its own, or you can use a variable within a mathematical function that you enter into the parameter text field instead. In this way, Rocky enables you to create dynamic relationships between parameters, and enables you to change and update placeholder values quickly. Using Input variables is also a good way to make complicated project setups simpler and easier to verify then by manually entering the same information into many different parameter field locations.

But because it can be hard to remember what parameters use what variables, Rocky uses the Expressions list to help you keep track of where the Input variables are being referenced-even if it is being used within a function or other variable definition. The Expressions list is also useful for verifying what units the variable or function is using for the particular location specified, and for changing the value or units of the variable later without having to locate it again in the Data Editors panel.

Setting Input Variables

Important:Verify that your field units are set the way you want before using or creating a variable-or function including a variable-within a field. Even though variables are saved in the Variables list without units, as soon as a variable-or function including a variable-is used within a parameter field, it takes on whatever units are currently set for that field and those units are then recorded for that specific usage in the Expressions list. Changing the units for the field after the variable or function have already been entered only converts the numerical display to reflect the new units; the original variable or function amount and units are still retained. However, you can change the units and value later by editing it in the Expressions list.

Input variables can be defined and used at any point during the setup portion of your simulation. (See also Enter Input Variables or Mathematical Functions as Parameter Values (p. 372).) Most text fields in Rocky support entering variables or mathematical functions as parameter variables, including three-part text fields.

Most text fields will also support the entering of pre-defined mathematical functions (e.g., *floor* and *pow*) and constants (e.g., *pi* and *e*) that come standard with Python-based programs, including Rocky. And as such, these particular functions and constants cannot be used when defining Input variables.

Using Input Variables with Ansys Workbench

Input variables are also useful when setting up Ansys Workbench-connected projects as you can use Workbench's Design Points functionality to replace Input Variables from Rocky with new values. When used together with Rocky's Output Variables, this functionality helps you more easily iterate upon simulation and setup components.

Note: To see walk-through examples of using Input Variables within Workbench-connected projects, refer to the following:

- Tutorial - Static Structural in the *Rocky Tutorial Guide*
- Tutorial - Transient Structural in the *Rocky Tutorial Guide*

Input Variables Parameter Definitions

Use the images and tables below to help you understand how to set your Input variables.

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

Figure 2.166: Example of entering a variable into a text field

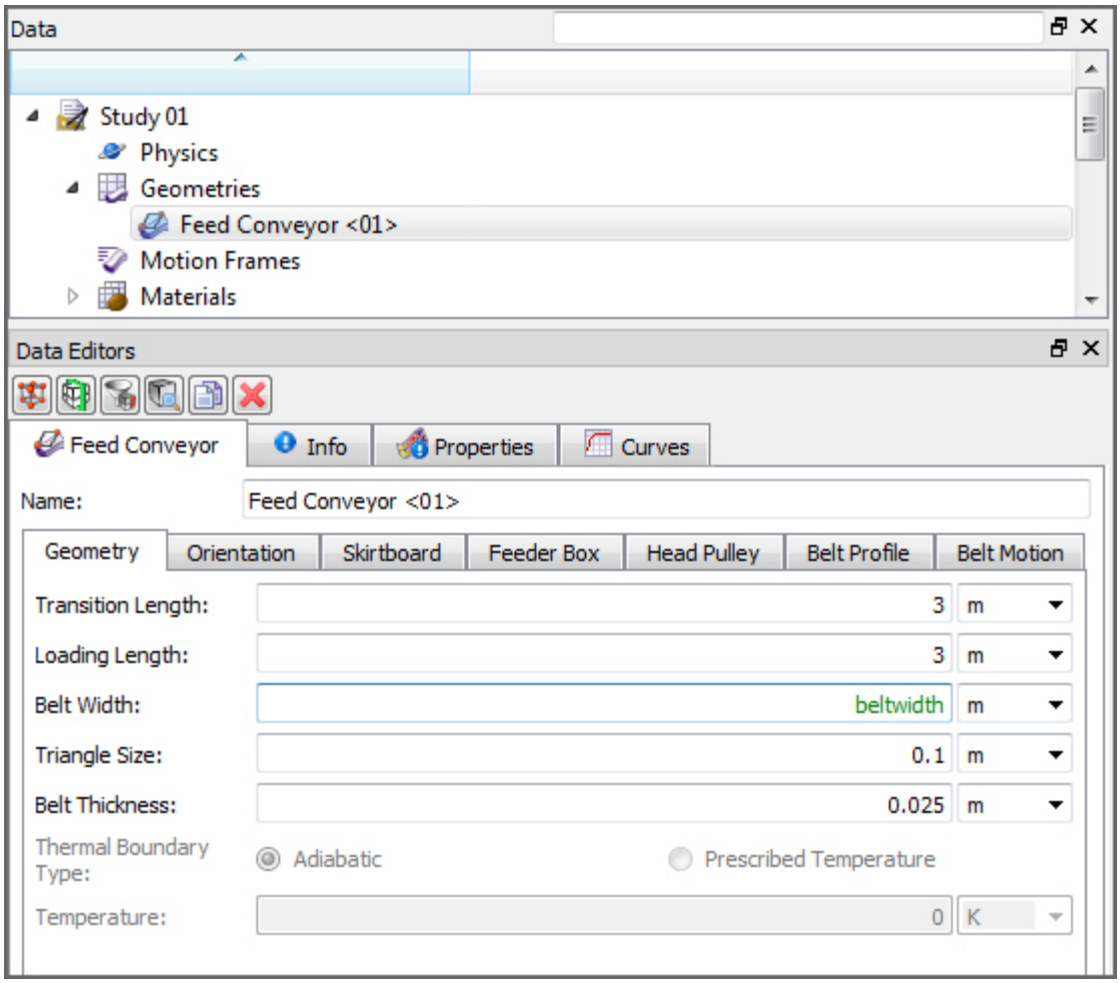


Figure 2.167: The variable name and units used for the field appears after double-clicking the field



Figure 2.168: The current value for the variable entered appears in the field after clicking away from the field



Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

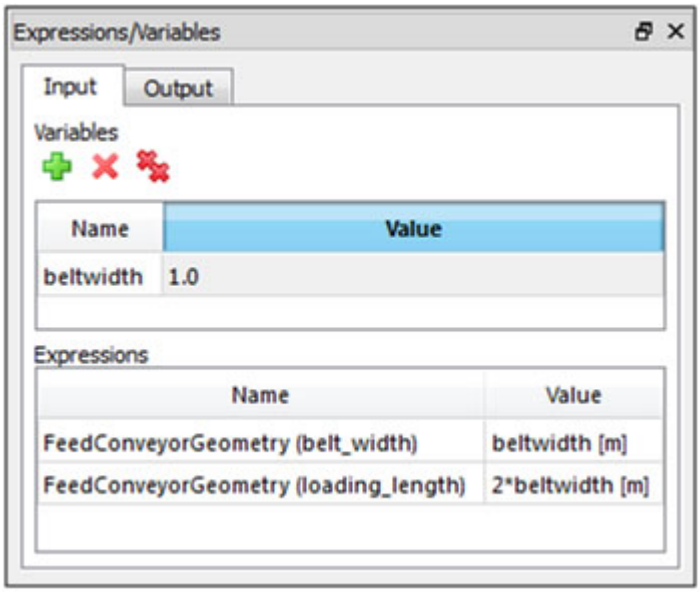


Figure 4: Expressions/Variables panel, Input tab showing one defined variable being used in two separate locations

Table 1: Expressions/Variables panel, Input tab options

Setting	Description	Range
Variables		
Name	Enables you to specify a unique identifier for the variable. **Tip:** Double-click the name to edit it. (See also Edit the Name of an Existing Input Variable (p. 248).)	The name must begin with a letter or underscore and can be followed by as many letters, underscores, or numbers as desired. Spaces and other symbols are not allowed. Letters are case sensitive.
Value	The numeric value-or a function including other existing variable(s)-assigned to the variable name. **Tip:** Double-click the value to edit it. (See also Edit the Value of an Existing Input Variable (p. 248).)	Any value; or any valid mathematical function including other existing variable(s), as long as the function does not create an infinite loop. (For more information about using functions, see Enter Input Variables or Mathematical Functions as Parameter Values (p. 372).)
Expressions		

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

Setting	Description	Range
Name	Lists the parameter field and location being used by the variable listed in the **Value** column.	Automatically provided. Cannot be edited.
Value	Lists the variable **Name** and the units it is currently using within the field specified by the Expression **Value** column.	Automatically provided. **Tip:** Double-click the value to edit the units. (See also Change the Units for an Input Variable-or Functions Using Input Variables-in Use by a Field (p. 249).)

What would you like to do?

- Learn more About Rocky and Ansys Workbench Integration (p. 144)
- Create a New Input Variable from within a Parameter Field (p. 247)
- Edit the Value of an Existing Input Variable (p. 248)
- Edit the Name of an Existing Input Variable (p. 248)
- Change the Units for an Input Variable-or Functions Using Input Variables-in Use by a Field (p. 249)
- Remove a Single Input Variable (p. 249)
- Remove All Input Variables (p. 249)

See Also:

- About Defining Output Variables (p. 250)
- Set Your Rocky Preferences (p. 226)

CREATE A NEW INPUT VARIABLE FROM WITHIN A PARAMETER FIELD

Note:

This method works only when using numerical values to define the variable. If for the variable definition, you want to reference other existing variable(s) within a function, you must define the new variable using the Expressions/Variables panel.

1. Ensure the units for the field are set the way you want. Important:Though the variable itself will be saved without units, its use in this particular field will be affected by the units that are displayed during the next step.

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

2. Directly into the field in which you want to use the variable, enter a name for the new variable, and then press Enter. Tips: The name must begin with a letter and must have no spaces. Only letters, numbers, and underscores are recognized (no symbols). Letters are case sensitive. The nme must not include pre-defined mathematical functions nor constants.

3. In the **Variable Creation** dialog, enter a numerical value for the variable, and then click **Create variable**. You will notice all of the following events:

   - The variable name is replaced by the numerical value in the parameter field. The variable is also using the units that were set for that field when the variable was entered. Tip: Double-click the field to see the full variable name and the units that it is using.

   - The new variable is displayed in the following two places on the **Input** tab of the **Expressions/Variables** panel (from the **Tools** menu, click **Expressions/Variables**):

     – Under **Variables**, where it is saved without units for use in other parameter fields. Tip: You can edit both the name and the value by double-clicking the information you want to change. (See also Edit the Name of an Existing Input Variable (p. 248) or Edit the Value of an Existing Input Variable (p. 248).)

     – Under **Expressions**, where the variable name and units applied in that particular usage are saved along with the location of its use. Tip: You can edit the value and units recorded here by double-clicking the information in the **Value** column, and then modifying anything before the closing bracket.

See Also:

- About Defining and Using Input Variables (p. 243)

EDIT THE VALUE OF AN EXISTING INPUT VARIABLE

1. From the **Input** tab on the **Expressions/Variables** panel (from the **Tools** menu, click **Expressions/Variables**), double-click the **Value** for the variable **Name** you want to change. The **Value** field for that variable becomes active.

2. Enter the new value you want and then press Enter. The new value is saved. **Notes**:

   - The variable value is always saved without units. When used in a field, this variable will use in that instance whatever units are currently being displayed for that field.

   - Once used in a field, you cannot change from the field the units being used by the variable in that particular field. (See also Change the Units for an Input Variable—or Functions Using Input Variables-in Use by a Field (p. 249).) You can, however, change the units by modifying the **Value** information in the **Expressions/Variables** panel under **Expressions**. Just double-click the **Value** you want to change, and then modify anything you want before the closing bracket.

See Also:

- About Defining and Using Input Variables (p. 243)

EDIT THE NAME OF AN EXISTING INPUT VARIABLE

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

1. From the **Input** tab on the **Expressions/Variables** panel (from the **Tools** menu, click **Expressions/Variables**), double-click the **Name** for the variable you want to change. The **Name** field for that variable becomes active.

2. Enter the new name you want and then press Enter. The new name is saved in the panel and is also replaced in any field making use of the variable.

See Also:

- About Defining and Using Input Variables (p. 243)

CHANGE THE UNITS FOR AN INPUT VARIABLE-OR FUNCTIONS USING INPUT VARIABLES-IN USE BY A FIELD

- Do one of the following:

  – In the **Expressions/Variables** panel under **Expressions**, find the instance of the variable whose units you want to change, double-click the **Value** for that usage, and then modify the units displayed in the brackets.

  – Remove the variable name-or function using the variable name-from the field, set the units for the field to the ones you want adopted by the variable or function, and then re-enter the variable name-or function using the variable name-again. **Tip**: Double-click the field to see the full variable name-or function using the variable name-and the units that it using in that instance.

See Also:

- About Defining and Using Input Variables (p. 243)

REMOVE A SINGLE INPUT VARIABLE

- From the **Input** tab of the **Expressions/Variables** panel, select the **Name** of the variable you want to remove, and then click the **Remove** button. Just that variable is removed from the list.

  Note: The last value replaces the variable name in any parameter field the removed variable was being used, and the **Expressions** list recording for that variable is cleared.

See Also:

- About Defining and Using Input Variables (p. 243)

REMOVE ALL INPUT VARIABLES

- From the **Input** tab of the **Expressions/Variables** panel, select a variable to enable the buttons, and then click the **Remove All** button. All the variables in the list on that tab are removed from the list.

  Note: The last values replace all the respective variable names in any parameter field the removed variables were being used, and the **Expressions** list recordings for the all the variables are cleared.

See Also:

- About Defining and Using Input Variables (p. 243)

DEFINING OUTPUT VARIABLES

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

Defining output variables enable you to distill a set of data points from a Property or Curve into a single final value.

What would you like to do?

- Learn About Defining Output Variables (p. 250)
- Define a New Output Variable (p. 254)
- Edit the Details of an Output Variable (p. 254)
- Remove a Single Output Variable (p. 254)
- Remove All Output Variables (p. 255)

See Also:

- About Defnining and Using Input Variables (p. 243)
- Set Your Rocky Preferences (p. 226)

ABOUT DEFINING OUTPUT VARIABLES

Output variables enable you to distill a set of data points from a Property or Curve into a single final value. (See also Use Properties and Curves (p. 931).) This process is illustrated in the example provided in Figure 1 below.

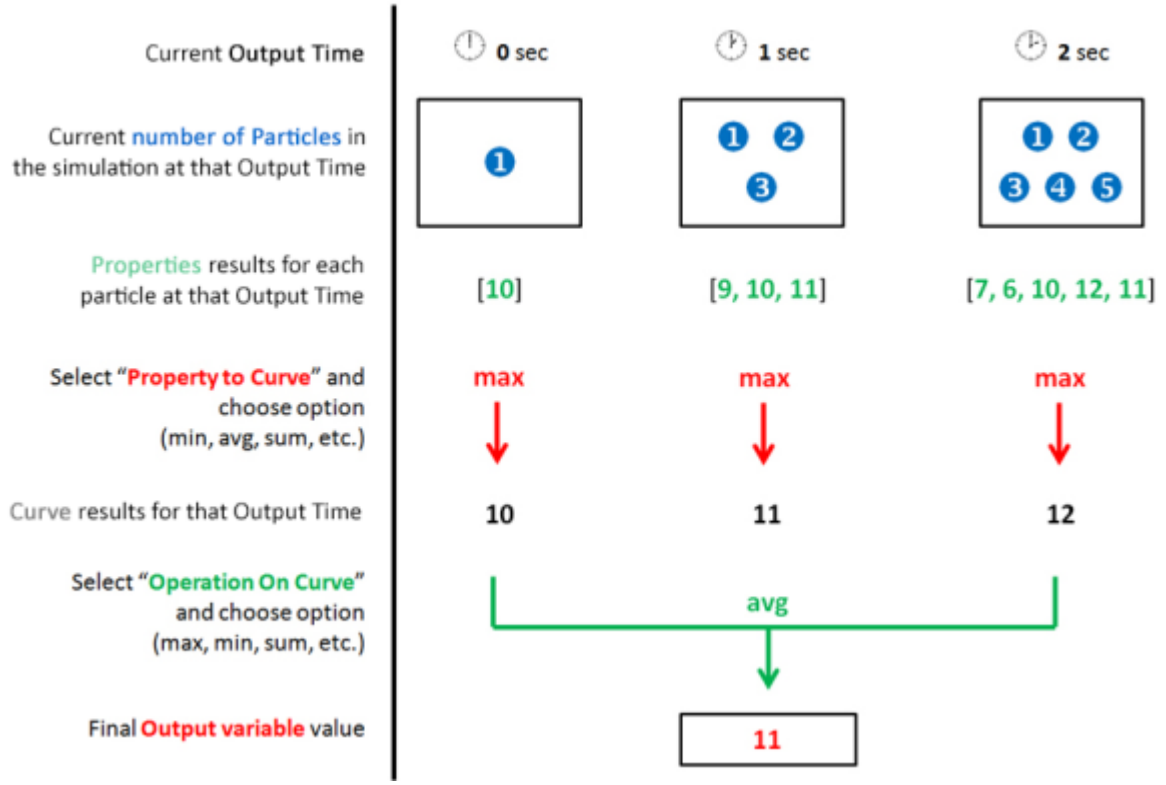


Figure 1 : Example of how Output variables are distilled into one value from a data set you define

In this example, the output time for the data set is limited to the first three seconds of the simulation, which then gives us the number of particles available at each of those three outputs. Then, the

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

Property value for each particle at each of the three outputs is provided. Those Property values are then distilled into a single-value Curve for each output time, in this case the maximum value out of each of the three Property data sets is selected. Out of the three resulting Curve values, one last distillation, in this case by average value, gives us the final value that will be saved to the Output variable, in this case, 11.

USING OUTPUT VARIABLES INSTEAD OF PLOTS

You can use Output variables as an alternative to generating and measuring data in Time plots. (See also About Graphing (Plot or Histogram) a Data Set Within Rocky (p. 1101)). Unlike a Time plot, an Output variable allows you to distill a set of Property or Curve values into a single value in one, easy-to-find location. For example, this method would be a benefit when needing to calculate the average mass of particles in a particular cube-shaped portion of the simulation between 3 and 10 seconds simulation time. (See also Filter Views and Data with User Processes (p. 1174).)

USING OUTPUT VARIABLES WITH ANSYS WORKBENCH

You can also use Output variables to distill data into a single value for exporting out of Rocky and into an Ansys Workbench project. When used together with Rocky's Input Variables, Workbench's Design Points functionality enables you to replace Input Variables from Rocky with new values, and then have Rocky provide the resulting Output value, all of which helps you more easily iterate upon simulation and setup components.

Tip: To see walk-through examples of using Output Variables within Workbench-connected projects, refer to the following Tutorials:

- Tutorial - Static Structural in the *Rocky Tutorial Guide*
- Tutorial - Transient Structural in the *Rocky Tutorial Guide*

OUTPUT VARIABLES PARAMETER DEFINITIONS

Use the images and table below to help you understand how to define Output variables.

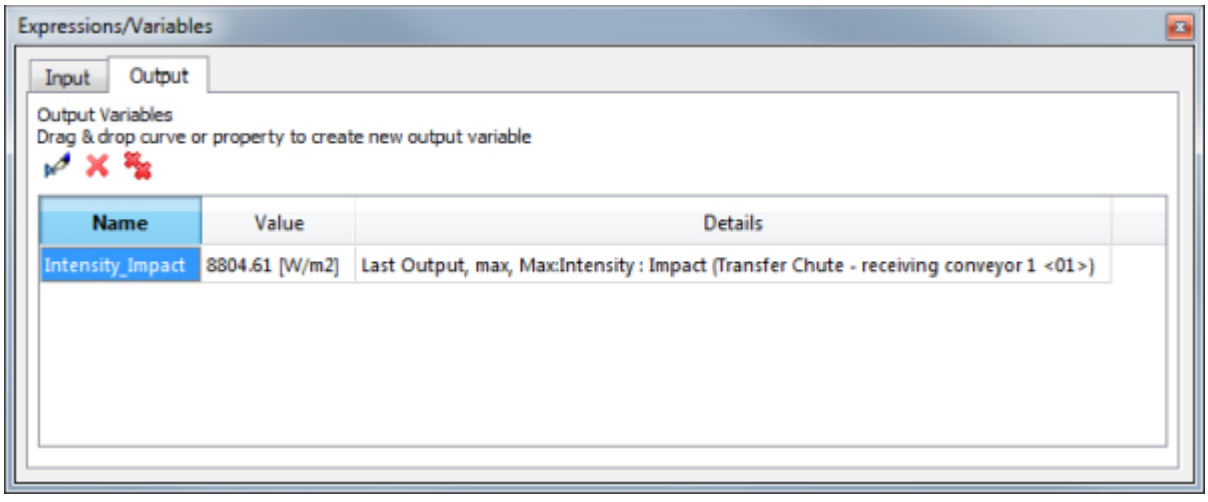


Figure 2: Expressions/Variables panel, Output tab showing one defined Property variable

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

Figure 2.169: Edit Properties dialog for Output variables

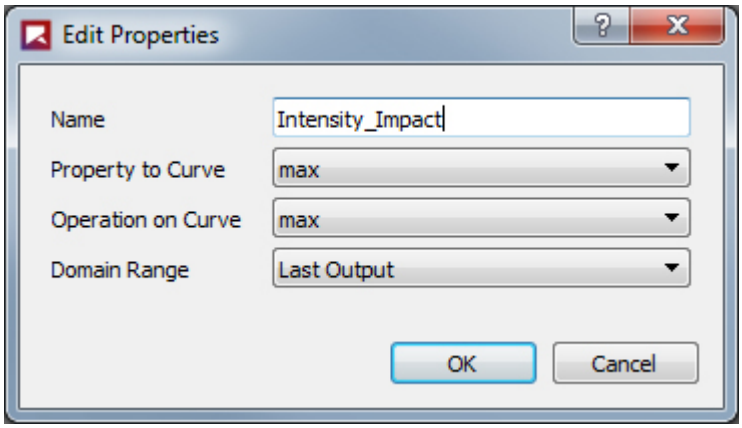


Table 1 : Expressions/Variables panel, Output tab and Edit Properties dialog options

Setting	Description	Range
Name	Enables you to specify a unique identifier for the variable. By default, it is the name of the Curve or Property that you have chosen but can be edited as desired.	The name must begin with a letter or underscore and can be followed by as many letters, underscores, or numbers as desired. Spaces and other symbols are not allowed. Letters are case sensitive.
Value	The numerical value assigned to the variable as determined by the Curve or Property that you have chosen and the constraints you have assigned.	Automatically calculated based upon the settings you have chosen.
Details	A summary of the constraints imposed upon the Curve or Property that results in the **Value** displayed.	Automatically listed based upon the settings you have chosen.
Property to Curve	When the variable is based upon a Property, this is the function that will convert the Property values at each output time in the **Domain Range** to a single value. This single value will then be acted upon by the **Operation on Curve** function to get the final **Value**.	Min; Max; Average; Sum; Sum Squared; Variance; Standard Deviation For more information, see Use Properties and Curves (p. 931).
Operation on Curve	If the variable is based upon a Curve, this is the operation that will be applied to it to generate the final **Value** for the variable. If the variable is based upon a Property, this is the operation that will be applied to the single value generated from the **Property to Curve** value to achieve a final **Value** for the variable.	Min; Max; Average; Sum; Sum Squared; Variance; Standard Deviation For more information, see Use Properties and Curves (p. 931).
Domain Range	Defines what Output Times of the simulation are included in the final **Value** calculation. The options are as follows:	Application Time Filter; All; Last Output; Time Range; Specify Time; After Time;

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

Setting	Description	Range
	• **Application Time Filter**: Use the range settings on the Time Filter dialog to limit the output times used. (For more information, see About the Time Toolbar (p. 1279).) • **All**: Use all available Output Times in the simulation. • **Last Output**: Use only the very last (most recent) output time that was calculated. • **Time Range**: Limit the output time values by a specific range that you choose. (Uses the **Initial** and **Final** values you set.) • **Specific Time**: Choose only a single specific output time. (Uses the **At Time** value you set.) • **After Time**: Use all output times available after a specific time period of your choosing. (Uses the **Initial** value you set.) • **Time Range Relative to Simulation End**: Specify how long before the very end of the simulation to include output times. (Uses the **Range from end** value you set.)	Time Range Relative to Simulation End
Initial	When **Time Range** or **After Time** is chosen for **Domain Range**, this is the starting time to begin calculations.	Any value between 0 and the final simulation time.
Final	When **Time Range** is chosen for **Domain Range**, this is the ending time when calculations are stopped.	Any value between the **Initial** time and the final simulation time.
At Time	When **Specific Time** is chosen for **Domain Range**, this is the exact moment in which the calculation will be performed.	Any value between 0 and the final simulation time.
Range from end	When **Time Range Relative to Simulation End** is chosen for **Domain Range**, this is the period of time before the final simulation time in which output times will be included in the calculation. For example, when you want to include only the last X seconds of the simulation.	Any value between 0 and the final simulation time.

What would you like to do?

- Learn more about About Rocky and Ansys Workbench Integration (p. 144)
- Define a New Output Variable (p. 254)
- Edit the Details of an Output Variable (p. 254)

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

- Remove a Single Output Variable (p. 254)
- Remove All Output Variables (p. 255)

See Also:

- About Defnining and Using Input Variables (p. 243)
- Set Your Rocky Preferences (p. 226)

DEFINE A NEW OUTPUT VARIABLE

1. Set up and process the simulation as you normally would.
2. Do all of the following: - Select the **Curve** or **Property** tab that contains the information you are interested in. (From the **Data** panel, select **Particles,** the geometry, or user process for which you want data and then in the **Data Editors** panel, select the tab for which you are interested.) - Click and drag the **Name** of the data you want to the **Output** tab of the **Expressions/Variables** panel. (From the **Tools** menu, click **Expressions/Variables**.) - Release the mouse. The data appears as a new variable in the **Output** list.

See Also:

- About Defining Output Variables (p. 250)
- Edit the Details of an Output Variable (p. 254)
- Remove a Single Output Variable (p. 254)
- Remove All Output Variables (p. 255)

EDIT THE DETAILS OF AN OUTPUT VARIABLE

1. From the **Output** tab of the **Expressions/Variables** panel, select the **Name** of the variable you want to edit, and then click the **Edit** button.
2. From the **Edit Properties** dialog, enter the information you want as explained in Table 1 (p. 250), and then click **OK**.

See Also:

- About Defining Output Variables (p. 250)
- Define a New Output Variable (p. 254)
- Remove a Single Output Variable (p. 254)
- Remove All Output Variables (p. 255)

REMOVE A SINGLE OUTPUT VARIABLE

- From the **Output** tab of the **Expressions/Variables** panel, select the **Name** of the variable you want to remove, and then click the **Remove** button. Just that variable is removed from the list.

See Also:

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

- About Defining Output Variables (p. 250)
- Define a New Output Variable (p. 254)
- Edit the Details of an Output Variable (p. 254)
- Remove All Output Variables (p. 255)

REMOVE ALL OUTPUT VARIABLES

- From the **Output** tab of the **Expressions/Variables** panel, select a variable to enable the buttons, and then click the **Remove All** button. All the variables in the list on that tab are removed from the list.

See Also:

- About Defining Output Variables (p. 250)
- Define a New Output Variable (p. 254)
- Edit the Details of an Output Variable (p. 254)
- Remove a Single Output Variable (p. 254)

2.6. Frequently Used Tasks

Use this section to quickly find and perform common tasks in Rocky.

What would you like to do?

- Enter Input Variables or Mathematical Functions as Parameter Values (p. 372)
- Change Set Up Parameters in a Partially Processed Simulation (p. 846)
- Change the Speed of a Default Conveyor (p. 373)
- Change Particle Adhesion (p. 374)
- Set Particle Size Ranges (p. 375)
- See Surface Wear on the Geometry Itself (p. 378)
- Import a Custom Particle Shape (p. 670)
- Use the 1-Way Fluent Method to Process Fluent and Rocky Simulations (p. 383)
- Use the 2-Way Fluent Method to Process Fluent and Rocky Simulations (p. 386)
- Enable Particle Breakage Calculations (p. 397)
- Create and View a Particle Assembly (p. 398)
- Enable Thermal Modeling Calculations (p. 400)

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

- Add a Logo to a 3D View (p. 402)
- Reuse 3D View Window, Workspace, and User Process Setups in Other Projects (p. 403)
- Undo or Redo Several Actions at Once (p. 108)
- Set Up a New GPU Card (p. 404)

See Also:

- Set Your Rocky Preferences (p. 226)

2.6.1. Enter Input Variables or Mathematical Functions as Parameter Values

You can create Input variables in Rocky (see Defining Your Variables (p. 243)) and then use them in place of actual numerical values in situations where you want more flexibility. These variables can be used alone or within functions.

Tip:

Rocky includes several visual feedback mechanisms, like text color, to help you ensure the variable names and mathematical functions you enter are correct. See Use Colored Text to Validate the Syntax of Your Entries (p. 117) for more information.

Important:Verify that your field units are set the way you want before using a variable-or function using a variable-within a field. Even though variables are always saved in the Variables list without units, as soon a variable-or function using a variable-is used within a parameter field, it takes on whatever units are currently set for that field and then those units are recorded for that specific usage in the Expressions list. Changing the units for the field after the variable or function have already been entered only converts the numerical display to reflect the new units; the original variable or function amount and units are still retained. However, you can change the units and value later by editing it in the Expressions list.

Most text fields in Rocky support entering variables or mathematical functions as parameter variables, including three-part text fields. There are a few exceptions to be aware of, however. (See also I cannot enter an input variable or mathematical function into a text field (p. 1334).)

Most of these parametric expressions will be retained while saving a copy of a project for restart purposes, but there are a few exceptions for some **Start Time** and **Stop Time** fields. (See also I get a "Links removed" message when I save my project for restart purposes (p. 1381).)

Most text fields will also support the entering of pre-defined mathematical functions (e.g., *floor* and *pow*) and constants (e.g., *pi* and *e*) that come standard with Python-based programs, including Rocky. And as such, these particular functions and constants cannot be used when defining Input variables.

What would you like to do?

- Replace a Parameter Value with a Variable (p. 372)
- Replace a Parameter Value with a Function Using a Variable (p. 373)
- Replace a Parameter Value with a Pre-Defined Mathematical Constant (p. 373)

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

See Also:

- View Frequently Used Tasks (p. 371)
- I cannot enter an input variable or mathematical function into a text field (p. 1334)

REPLACE A PARAMETER VALUE WITH A VARIABLE

- Using the **Data** and **Data Editors** panels, locate and select the field for the parameter you want to replace, exactly enter the name of the variable you want to use, and then press Enter. The current value set for the variable in the **Expressions/Variables** panel is displayed in the field.

Tip:

To see the units that are set for the variable you used, double click the value in the field. The units are displayed in brackets [ ] after the variable name.

Note:

You cannot use pre-defined functions or constants to define your variable.

See Also:

- Enter Input Variables or Mathematical Functions as Parameter Values (p. 372)
- Replace a Parameter Value with a Function Using a Variable (p. 373)
- Replace a Parameter Value with a Pre-Defined Mathematical Constant (p. 373)

REPLACE A PARAMETER VALUE WITH A FUNCTION USING A VARIABLE

- Using the **Data** and **Data Editors** panels, locate and select the field for the parameter you want to replace, and then enter the function you want to use.

See Also:

- Enter Input Variables or Mathematical Functions as Parameter Values (p. 372)
- Replace a Parameter Value with a Variable (p. 372)
- Replace a Parameter Value with a Pre-Defined Mathematical Constant (p. 373)

REPLACE A PARAMETER VALUE WITH A PRE-DEFINED MATHEMATICAL CONSTANT

- Using the **Data** and **Data Editors** panels, locate and select the field for the parameter you want to replace, and then enter the pre-defined mathematical function or constant you want to use.

See Also:

- Enter Input Variables or Mathematical Functions as Parameter Values (p. 372)
- Replace a Parameter Value with a Variable (p. 372)

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

- Replace a Parameter Value with a Function Using a Variable (p. 373)

2.6.2. Change the Speed of a Default Conveyor

You can change the speed of a default receiving or feed conveyor at any point during the simulation. For example, you might want to measure reclaiming belt power by starting off a simulation with a feed conveyor stopped until a hopper is filled with particles, and then ending the simulation with the conveyor up to full speed.

TO CHANGE THE SPEED OF A DEFAULT CONVEYOR:

1. Ensure the default conveyor for which you want to change belt speed has already been added. (See also Add and Edit Geometry Components (p. 480).)

2. From the **Data** panel, under **Geometries**, select the **Receiving Conveyor** or **Feed Conveyor** for which you want to change belt speed. The parameters for that geometry are displayed in the **Data Editors** panel.

3. From the **Data Editors** panel, on the **Receiving Conveyor** or **Feed Conveyor** tab, ensure the **Belt Motion** sub-tab is selected, and then do all of the following:

    a. In the **Belt Speed** box, enter the full speed you want the belt to reach after acceleration is complete.

    b. In the **Beginning Start Time** box, enter the amount of time that you want the simulation to run before the belt starts accelerating.

    c. In the **Acceleration Period** box, enter the amount of time you want the belt to increase from full stop to full **Belt Speed**.

    d. In the **Beginning Stop Time** box, enter the amount of time that you want the simulation to run before the belt starts decelerating.

    e. In the **Deceleration Period** box, enter the amount of time that you want the belt to decrease from full **Belt Speed** to full stop.

See Also:

- View Frequently Used Tasks (p. 371)

2.6.3. Change Particle Adhesion

You change particle adhesion when you want a particle to act wetter or "stickier" than normal during a simulation. This might be useful in showing how ore flows through handling equipment during the wet season as compared to the dry, for example.

Tip: It is good practice to calibrate your particle settings to the real-world behavior of the material you are simulating before changing settings like adhesion. You may wish to use the Material Wizard and/or the Calibration Suite to help you with this task.

TO CHANGE PARTICLE ADHESION:

1. Set up the simulation as you normally would. (See also Setting Up a Simulation (p. 411).)

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

2. During the Set Simulation-Wide Parameters step, ensure you select both the **Rolling Resistance Model** and the **Adhesive Force** you want from the **Physics** | **Momentum** tab. (See also About Physics Parameters (p. 417).)

3. Adjust Materials Interactions values by doing all of following:

   a. From the **Data** panel, select **Materials Interactions**.

   b. From the **Data Editors** panel, choose the materials you want to modify from the **Select Materials** lists.

   c. Adjust one or all of the settings provided including friction, adhesive distance, and force and/or stiffness fractions. (See also About Modifying Materials Interactions and Adhesion Values (p. 597).)

4. Adjust Rolling Resistance values by doing all of the following:

   a. From the **Data** panel, under **Particles**, select the rigid, convex Particle set (p. 1393) for which you want to change adhesion.

   b. From the **Data Editors** panel, select the **Movement** tab, and then adjust the **Rolling Resistance** value. **Notes**: - The **Type C: Linear Spring Rolling Limit** Rolling Resistance model is recommended for sticky particles. (See also About Physics Parameters (p. 417).) - Rolling Resistance is not usually needed for shaped particles (p. 1395).

5. Process the simulation and then continue to adjust the Materials Interactions and Rolling Resistance values until you get the results you want.

See Also:

- View Frequently Used Tasks (p. 371)

2.6.4. Set Particle Size Ranges

You set a particle size range-otherwise known as a Particle Size Distribution (PSD)-when you want particles of one shape but different sizes to be used in the simulation. If you set more than one size for a Particle set, Rocky uses a calculation to specify the distribution of sizes within any two dimensions you specify. This helps to more accurately reflect real matter.

When setting size ranges, the following three methods, or **Size Types**, can be used as a basis for defining your particle sizes (Figure 1):

- **Sieve Size** (formerly "Sieve"), which for especially default Rocky shapes, bases particle size upon a virtual mesh hole just big enough for the particle to pass through. **Note**: Due to the way Sieve Size is calculated, it should be used only for even and balanced particle shapes (such as the default **Solid** particles included with Rocky) and not for fibers or shapes with high aspect ratios. (For more detail on how Sieve Size is calculated in Rocky, see the DEM Technical Manual. (From the Rocky **Help** menu, point to **Manuals**, and then click **DEM Technical Manual**.))

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

- **Equivalent Sphere Diameter** (formerly "Equivalent Diameter"), which for especially irregular objects, bases particle size upon the diameter of a sphere with equivalent volume.

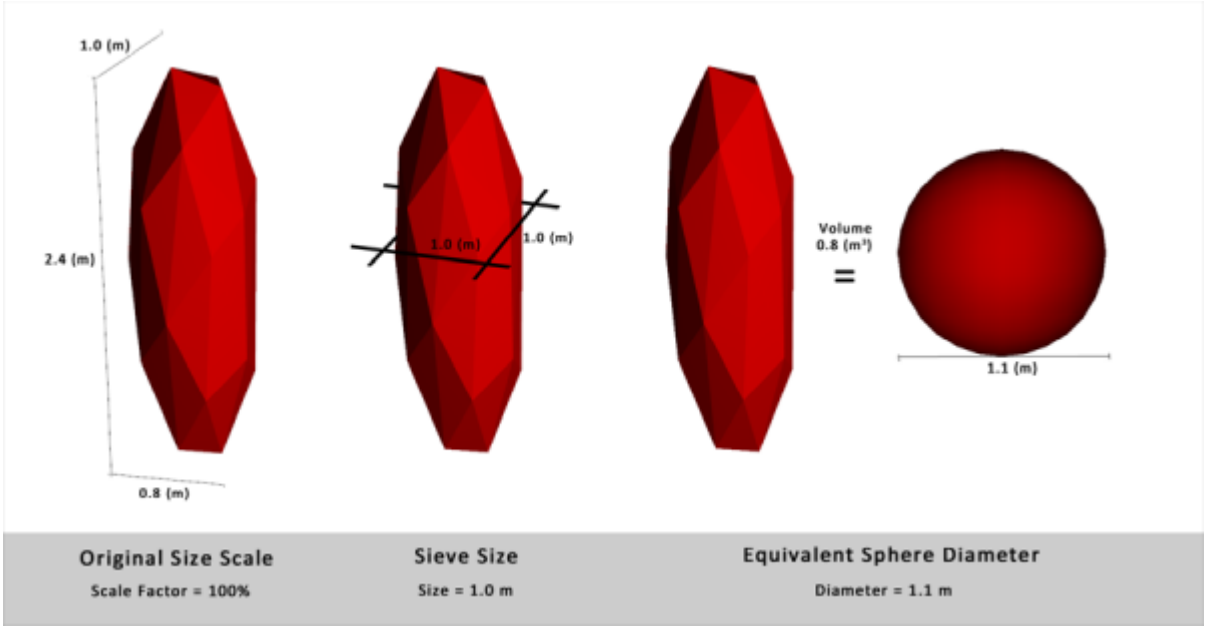
Tip:

To see a walk-through example of setting and using Equivalent Sphere Diameter, refer to the following Tutorial: Tutorial - Windshifter in the *Rocky Tutorial Guide*. **Original Size Scale**, which for especially imported shapes, bases particle size upon the original size of the shape.

Tip:

To see a walk-through example of setting and using Original Size Scale, refer to the following Tutorial: Tutorial - Tablet Coater in the *Rocky Tutorial Guide*.

Figure 2.170: Comparison of the three different Size Types in Rocky



The PSD in Rocky is linear by Particle Mass or, on a semi-log scale, by Particle size and not by number. When you enter the percentage of cumulative mass and then either particle sieve **Size**, sphere **Diameter**, or original size **Scale Factor** (representing the three available **Size Types**, respectively), these represent points on a semi-log plot. Anything between two points follows the linear rule:

$$Mass = A + B * \log(\text{Size})$$
$$Mass = A + B * \log(\text{Diameter})$$
$$Mass = A + B * \log(\text{Scale Factor})$$

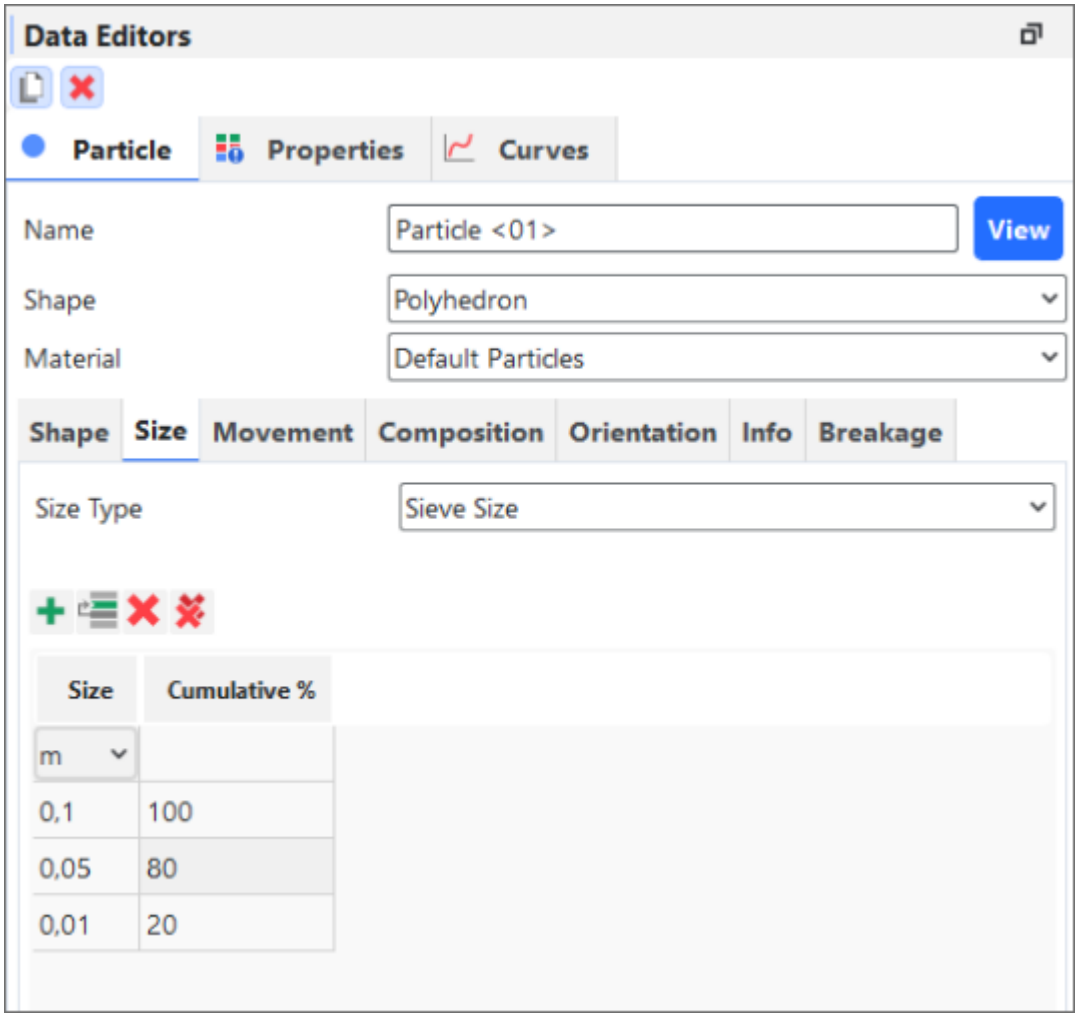
(See also About Adding and Editing Particle Sets (p. 602).)

For example, if you specify 0.1 and 0.05 (m) particle sizes measured by sieve size, you will get a range of various particle sizes within those two dimensions.

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

Use the image and procedure below to set size ranges for your Particle sets.

Figure 2.171: Example Size Distribution Settings for Sieve Size



TO SET A PARTICLE SIZE RANGE OR PSD:

1. Ensure that the Particle set for which you want to specify size ranges has been added. (See also Add a New Particle Set (p. 678).)

2. From the **Data** panel, select the Particle set you want (for example, Particle <01>) and then from the **Data Editors** panel, select the **Size** tab.

3. Set what you want for **Size Type**. (See also About Adding and Editing Particle Sets (p. 602).)

4. Click the **Add** button once for each additional size range you require. **Note**: For rigid particles composed of Single Elements, there is no limit to the amount of rows you can add. For flexible or rigid particles composed of Multiple Elements (also known as *meshed* (p. 1392) particles-see the **Composition Tab** section in the About Adding and Editing Particle Sets (p. 602) topic), only one row can be defined per Particle set.

5. Under **Size**, **Diameter**, or **Scale Factor** respectively, and starting with the first row, enter the maximum and minimum size values for each range in decreasing order. *For example*, to get a range of sieve sizes between 0.05-0.1 m and 0.01-0.05 m, enter 0.1, 0.05, and 0.01 in the first,

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

second, and third rows respectively. (See Figure 2 above for example.) **Note**: The smallest value you enter (last row) will also be shown at that exact size (no range).

6. Under **Cumulative %**, starting with the second row, indicate what percentage of the total particle mass you want of that size value range by entering in decreasing order the previous row's percentage minus the percentage you want. *For example*, if you want 20% of the 0.05-0.1 m range, enter 80 (100-20); to get 60% of the 0.01-0.05 m range, enter 20 (80-20). (See Figure 2 above for example.)

Note: The percentage indicated in the final row also indicates what particle mass amount will be of that exact size (no range). For example, if the smallest size you enter is 0.01 m and you have entered 20 for **Cumulative %**, 20% of the particle mass will be exactly 0.01 m in size in addition to the ranges you have set.

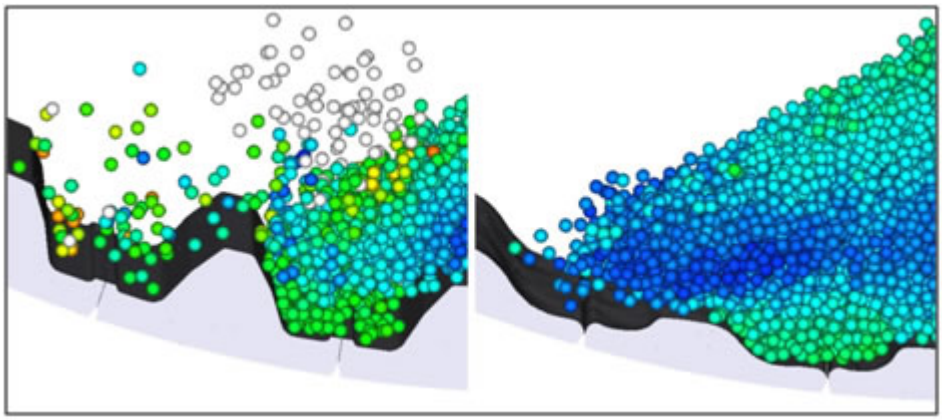
See Also:

- View Frequently Used Tasks (p. 371)

SEE SURFACE WEAR ON THE GEOMETRY ITSELF

There are two primary ways you can use Rocky to gain an understanding of how your geometries will wear over time. One way is by enabling surface wear modification, which changes the physical appearance of the geometry as the simulation progresses (Figure 1).

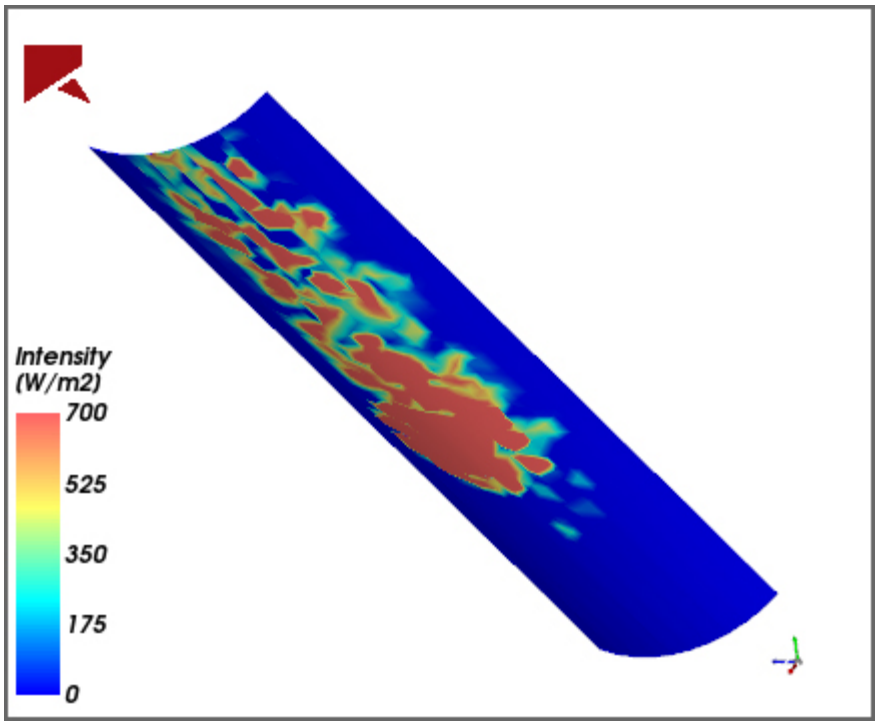
Figure 2.172: Surface wear modification



A second way is to turn on Boundary Collision Statistics (see also About Collision Statistics for Boundaries (p. 312) and then view a color map of the surface intensity (Figure 2), impact velocity, stresses, and more.

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

Figure 2.173: Color map of surface intensity



These features are independent of each other and are enabled from different areas of the Rocky UI. However, to view either type of wear data, you need to ensure that you have enabled enough geometry triangles prior to processing your simulation. Too few triangles will result in a jagged or blocky representation rather than a smooth one, but too many triangles can slow down your simulation.

Tip: To see a walk-through example of surface wear modification being using to analyze a SAG mill, review Tutorial - SAG Mill in the *Rocky Tutorial Guide*.

What would you like to do?

- View Surface Wear Modification on an Imported Geometry (p. 379)
- View a Color Map of Wear on the Default Belt or Imported Geometry Itself (p. 381)
- Learn more About Collision Statistics for Boundaries (p. 312)

See Also:

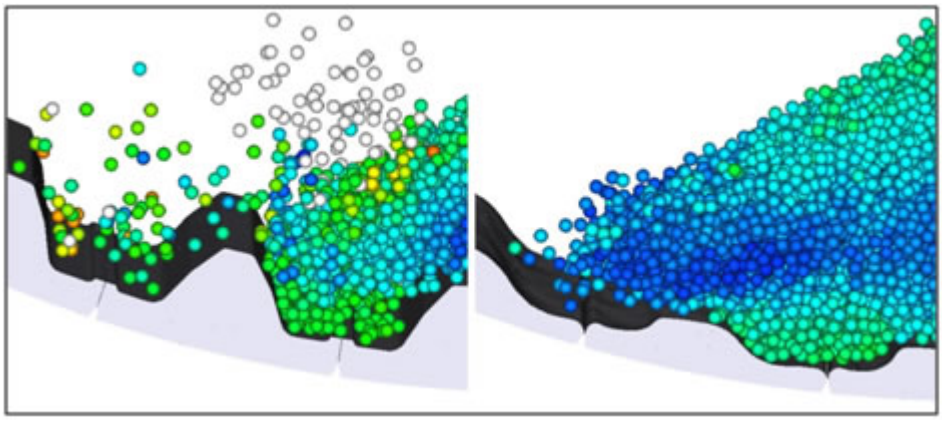
- View Frequently Used Tasks (p. 371)
- Understand Data Collection in Rocky (p. 306)

ENABLE AND VIEW SURFACE WEAR MODIFICATION ON AN IMPORTED GEOMETRY

One way you can use Rocky to gain an understanding of how your geometries will wear over time is by enabling surface wear modification. Turning this feature on for an imported geometry prior to processing changes the physical appearance of the geometry as the simulation progresses (Figure 1).

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

Figure 2.174: Surface wear modification



To view this type of wear data, you must ensure that you have enabled enough geometry triangles prior to processing your simulation. Too few triangles will result in a jagged or blocky representation rather than a smooth one, but too many triangles can slow down your simulation.

It is important to realize that this data is only collected after the Wear **Start** value on the **Solver** | **Time** tab is reached. This is to allow enough time for the particle flow within your simulation to reach a steady state before surface wear modification calculations begin.

Tips: To see walk-through examples of surface wear modification, refer to the following Tutorials:

- Tutorial - SAG Mill in the *Rocky Tutorial Guide*
- Tutorial - High Pressure Grinding Roll (HPGR) in the *Rocky Tutorial Guide*

TO ENABLE AND VIEW SURFACE WEAR MODIFICATION:

1. Set up the simulation as you normally would. (See also Set Simulation Parameters (p. 414)). **Tip**: For easier comparison after the simulation is complete, consider including a copy of the imported geometry that will remain unworn. (See also Duplicate a Data Panel Item (p. 846).)
2. Before processing the simulation, do all of the following:
    a. From the **Data** panel, under **Geometries**, select the imported geometry for which you want to enable surface wear. The parameters for the geometry become active in the **Data Editors** panel.
    b. From the **Data Editors** panel, on the **Geometry** tab, do all of the following:
        i. From the **Geometry** sub-tab, ensure **Triangle Size** is small enough to enable the wear detail you want. (0.1 m is recommended for most chutes and mills).
        ii. From the **Wear** sub-tab, select the **Wear Model** you want to use, and then enter the **Volume/Shear Work Ratio**. Note: This is a calibration step that comes from real-world wear data you have collected on similar types of equipment.
    c. From the **Solver** | **Time** tab, ensure that your Wear **Start** time is set to when you want wear data to be collected. (From the **Data** panel, select **Solver** and then from the **Data Editors** panel, locate this value on the **Time** sub-tab.) (See also About Solver Parameters (p. 464).) **Tip**:

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

It is best practice to set your Wear **Start** time to begin after a steady state has been reached for your particle flow.

3. Process the simulation as you normally would. (See also Processing a Simulation (p. 907).)

4. From a 3D View window (see also About 3D View Windows (p. 1040)), observe the changes to the surfaces of the geometry as the simulation progresses. **Tip**: If you included a copy of the original geometry in the simulation, you can use the unworn geometry as a comparison to one worn by surface wear modification.

Tips:

- If you want to analyze the modified geometry outside of Rocky, you can choose to export it to an STL file. (See also Export a Geometry Component to an STL File (p. 520).)

- It is possible to save a copy of an in-process simulation with surface wear and have the wear calculations continue in the copy. (See also Save a Copy of a Partially Processed Simulation for Restart Purposes (p. 855).) While you are unable to turn off wear calculations from the copy, if you no longer need wear, you can get around this by setting **Volume/Shear Work Ratio** to zero.

- It is also possible to have wear calculations be made on replicated geometries.

See Also:

- About Collecting Data in Rocky (p. 306)

- Enable and View Collision Statistics for Boundaries (p. 318)

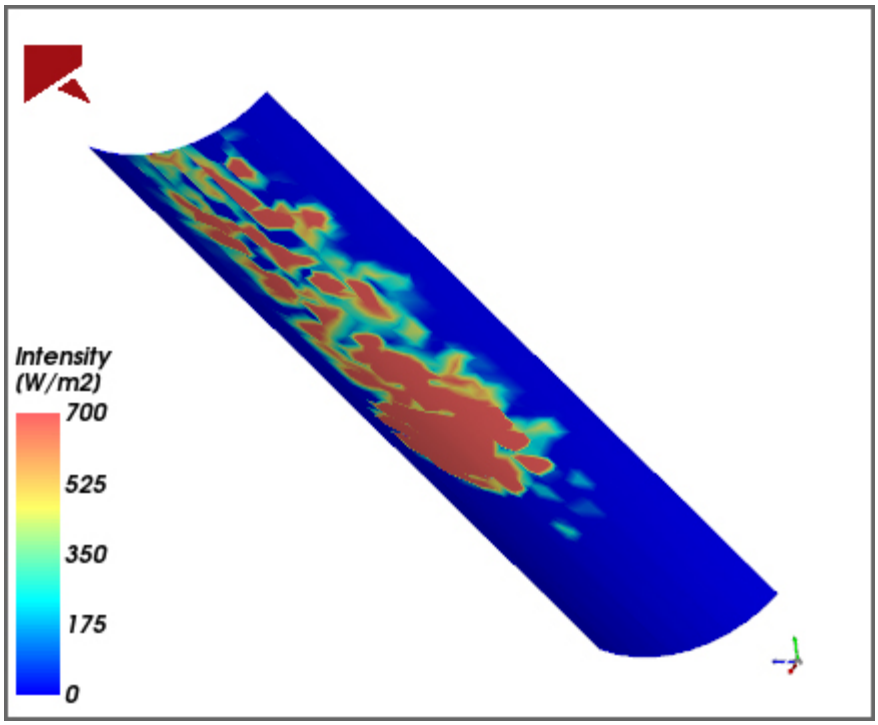
- Export a Geometry Component to an STL File (p. 520)

VIEW A COLOR MAP OF WEAR ON THE DEFAULT BELT OR IMPORTED GEOMETRY ITSELF

When you choose to turn on Boundary Collision Statistics (see also About Collision Statistics for Boundaries (p. 312) and enable wear-related categories such as **Intensities**, you are able to create and view a color map of the surface intensity (Figure 1), impact velocity, stresses, and more. These color maps can help you analyze the causes of excess wear on your conveyor belts or imported geometries.

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

Figure 2.175: Color map of surface intensity



TO VIEW A COLOR MAP OF WEAR ON THE DEFAULT BELT OR IMPORTED GEOMETRY ITSELF

1. Set up the simulation as you normally would. (See also Set Simulation Parameters (p. 414).)
2. Before processing your simulation, ensure all of the following:
   a. You have enabled the Boundary Collision Statistics that you want to be collected. (See also Enable and View Collision Statistics for Boundaries (p. 318).)
   b. You have verified that the default conveyor or imported geometry has its **Triangle Size** set small enough to enable the wear detail you want. (0.1 m is recommended for most chutes and mills). (From the **Data** panel under **Geometry**, select the component you want to verify. From the **Data Editors** panel, on the **Geometry** sub-tab, verify the **Triangle Size** value.)
3. Process the simulation as you normally would. (See also Processing a Simulation (p. 907).)
4. Ensure you have a 3D View displayed in the Workspace. (See also Create and Modify a 3D View (p. 1040).)
5. When processing is complete enough to show the wear you want to visualize, from the first drop down list on the **Coloring Settings** toolbar, choose the data you want to display (e.g., **Intensity**.) The geometries in your simulation change color and a color legend is displayed in the 3D View.
6. Do any or all the following to adjust the view and color map details:
   - To better focus on the area you want to view, use the mouse to pan, rotate, and zoom. (See also About Using the Mouse and Keyboard to Change the View (p. 1043).)

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

- To hide all items but the belt or geometry you want to focus on, use the eye icons to the left of the geometries and particles in the **Data** panel. (See also Show/Hide Components by Using Eye icons and Checkboxes (p. 113).)
- To modify the color bar, right-click the color legend and change the options listed. (See also About Color Scales (p. 1086).)
- To focus on a different time period within the simulation, use the arrows or slider on the Time toolbar. (See also About the Time Toolbar (p. 1279).)

See Also:

- Enable and View Collision Statistics for Boundaries (p. 318)
- Enable and View Surface Wear Modification on an Imported Geometry (p. 379)
- About Collision Statistics for Boundaries (p. 312)
- Understand Data Collection in Rocky (p. 306)

2.6.5. Use the 1-Way Fluent Method to Process Fluent and Rocky Simulations

There are five main steps to using the 1-Way Fluent CFD Coupling method to process a 1-Way coupled Rocky and Ansys Fluent simulation, as described below.

Tips:

- You may also complete this process from within Ansys Workbench. To accomplish this with transient Fluent data, however, there are separate steps you must follow. (See also Set Up and Run a 1-Way Fluent Transient Project within Ansys Workbench (p. 789).)
- You may also complete this process without using the Fluent UI. (See also Use a Journal to Run Fluent and Export Transient 1-Way Results (p. 1313).)

STEP I: VERIFY THAT ALL ROCKY-FLUENT PROGRAM REQUIREMENTS ARE MET

1. Before beginning Step II below, ensure all of the following program requirements have been met:
   - A Rocky-supported version of Ansys Fluent is installed upon a machine that also contains a Rocky program installation. (See also System Requirements (p. 67).) Note: Rocky does not need Fluent to be installed on the same machine in order to run a 1-Way Fluent simulation. However, Fluent needs Rocky to be installed on the same machine in order for Rocky to enable the **Rocky Export** menu item in Fluent.
   - If you are not certain whether you have installed the **Ansys Fluent Coupling Support** component during Rocky installation, first follow the Install Ansys Coupling Components (p. 792) procedure before beginning Step II.

STEP II: SET UP, PROCESS, AND THEN EXPORT THE CFD SIMULATION

1. In Ansys Fluent, set up the CFD simulation that you want to couple with Rocky. Important: Ensure that you complete all of the following; these exact settings are required for 1-Way Fluent coupled simulations:

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

a. From the **Fluent Launcher** dialog, do the following:

   i. Under **Dimension** ensure **3D** is selected.

   ii. Under **Options** select **Double Precision**.

b. From the **Solution Methods** tab, under **Pressure-Velocity Coupling Scheme**, ensure **SIMPLE** is selected.

Optional Steps:

- If you want to use any standard Fluent post processing options, set them up now before processing the simulation.
- If you want both sides of your coupled simulation to share identical geometries, set up those geometries now in Fluent, but be sure you save your Fluent Case File. Later in Rocky, you can choose to import those geometries directly from the CAS, CAS.H5, or CAS.GZ file that you will save from Fluent in the next step.
- If you want both sides of your coupled simulation to share the same geometry movements, set up the translation or rotation movements now using Fluent Moving Meshes. These movements will later be imported (with the F2R file) into Rocky (Step IV.2 below), which will automatically convert them into Rocky Motion Frames. Note: If you chose to define your moving meshes in Fluent using expressions, you will need to set up your Motion Frames in Rocky manually. (See also Step IV.2 below and About Creating and Applying Motion Frames to Imported Geometries (p. 533).)

2. From the **File** menu, **Write** the case to a location of your choosing. Tip: Remember the location of this directory as you will need to access files saved to it in later steps.
3. (Optional) Configure the options for Rocky Export by doing any or all of the following:
   - Define the directory you want to export by doing the following:

     a. From the Fluent **Rocky Export** menu, point to **Configure one-way export** and then click **Select Directory to Export**.

     b. From the **Select folder to write fluent to rocky files**, select the folder to where you want your fluid data saved, and then click **Select Folder**. Note: If you do not define this, Fluent will automatically use the same folder as your case file.

   - Define the additional variables you want to export by doing the following: Note: These additional variables can include static pressure, species variables such as mole fraction, diffusivity, and user defined memories. Other variables such as Velocity, Pressure Gradient, Temperature, and Fluid properties are always exported no matter what you select here.

     a. From the Fluent **Rocky Export** menu, point to **Configure one-way export** and then click **Select Variables to Export**. Note: To do this task, you must first have variables defined within Fluent.

     b. From the **Additional Variables Selector** dialog that appears, select the variables that you want included in the exported files, and then click **OK**.

   - Define how frequently you want files to be saved during the export by doing the following:

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

a. From the Fluent **Rocky Export** menu, point to **Configure one-way export** and then click **Set Output Frequency**.

b. From the **Fluent to Rocky (F2R) time-step multiplier** dialog define what you want for the **F2R Time Step Multiplier** value, and then click **OK**. Note: Only whole numbers are valid multipliers.

4. Export the data you want by doing one of the following:

   - To export steady-state data, do the following:

     a. Process the CFD simulation until the fluid flow reaches a steady state.

     b. From the Fluent **Rocky Export** menu, point to **Export one-way data** and then click **Export current data to Rocky**. Tips:

        – If you do not see this menu in Fluent, then first follow the Install Ansys Coupling Components (p. 792) procedure before continuing.

        – To perform this step before the Fluent case is done processing, stop Fluent from processing the case further before using **Rocky Export** menu.

          An F2R and other files needed for coupling is saved to the same folder as your Fluent case file (or the alternate folder you defined in step 3).

   - To export transient data, do the following:

     a. Begin the export process by doing one of the following:

        – To start recording from the initial conditions (without any prior Fluent results), initialize the Fluent case.

        – To start recording some time after Fluent results have been collected, process the CFD simulation until the fluid flow reaches the point that you want to start recording data, and then stop Fluent from processing the case further.

     b. From the Fluent **Rocky Export** menu, point to **Record one-way transient data**, and then click **Start one-way transient export**. Tip: If you do not see this menu in Fluent, then first follow the Install Ansys Coupling Components (p. 792) procedure before continuing.

     c. Begin (or resume) processing in Fluent until the point you want to stop recording data. Stop Fluent from processing the case.

     d. From the Fluent **Rocky Export** menu, point to **Record one-way transient data**, and then click **Stop one-way transient export**. An F2R and other files needed for coupling is saved to the same folder as your Fluent case file (or the alternate folder you defined in step 3).

5. Click **OK** to close the confirmation message dialog.

6. Save the Fluent case again.

STEP III: SET UP THE DEM SIMULATION IN ROCKY

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

1. In Rocky, set up your simulation project as you normally would. (See also Set Simulation Parameters (p. 414).) Tips:

   - If you want to use the exact same geometries as in the Fluent simulation, you may choose to import the CAS, CAS.H5, or CAS.GZ file you generated in Step II, which will include all available geometries as separate components.

   - If you enabled Energy Equations in the Fluent case, then ensure you also enable Thermal Modeling settings in Rocky. (See also Enable Thermal Modeling Calculations (p. 400).)

STEP IV: APPLY 1-WAY FLUENT COUPLING

1. From the **Data** panel, click **CFD Coupling** and then from the **Data Editors** panel, choose **Fluent (Fluid → Particle)** from the **Coupling Mode** list.

2. From the **Select Fluent 2 Rocky export file** dialog, locate and select the F2R file you exported in Step II, and then click **Open**. A new **1-Way Fluent** item appears beneath **CFD Coupling** in the **Data** panel. Important:For F2R files that include transient data, this step could take several minutes to complete. In addition, if you chose to set up Moving Meshes in Fluent, one new Frame will appear under **Motion Frames** in the **Data** panel for each Cell Zone with motions you have defined. Important:If you have defined your Moving Meshes in Fluent using expressions, Rocky will not be able to create the associated Motion Frames for you automatically. Rather, you must manually create your own Motion Frames in Rocky to match the Fluent motions as closely as possible. Tip: To save loading time, the Nodes for the imported fluid are turned off by default in the 3D View. To see them, from the Coloring tab, enable the **Nodes** checkbox. (See also About Using the Coloring Tab to Change a 3D View.)

3. Under **CFD Coupling**, select the new item and then from the **Data Editors** panel, on the **1-Way Fluent** tab, do the following:

   a. Set the **Start Time** you want.

   b. From the **Interactions** tab, do the following:

      i. From the **Particle** list, select (or multi-select) the Particle set name(s) for which you want to define interactions, and then select the various CFD laws you want for the selected set(s).

      ii. Select what you want for **Turbulent Dispersion**.

(See also About Using the 1-Way Fluent Method (p. 775).)

STEP IV: PROCESS THE COUPLED SIMULATION

1. In Rocky, process the simulation as you normally would. (See also Start Processing a Simulation from the Beginning (p. 913).)

See Also:

- About Using the 1-Way Fluent Method (p. 775)
- Set Up and Run a 1-Way Fluent Transient Project within Ansys Workbench (p. 789)
- Use a Journal to Run Fluent and Export Transient 1-Way Results (p. 1313)

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

2.6.6. Use the 2-Way Fluent Method to Process a Fluent and Rocky Simulation

There are five main steps to using the 2-Way Fluent method to process a coupled Rocky and Ansys Fluent simulation, as described below.

Note: Both Single phase (p. 1395) and Multiphase (p. 1392) approaches are covered in this procedure.

STEP I: VERIFY THAT ALL ROCKY-FLUENT PROGRAM REQUIREMENTS ARE MET

1. Before beginning Step II below, ensure all of the following program requirements have been met:

   - A Rocky-supported version of Ansys Fluent is installed upon the same machine that contains your Rocky program installation. (See also System Requirements (p. 67).)

   - If you desire to run Fluent as a parallel process (rather than a serial one), then the machine upon which you intend to run the Rocky portion of your coupled simulation must also run at least the host part of the Fluent process. (For more information about how the processing architecture works in Fluent, refer to your Ansys Fluent documentation.)

   - If you are not certain whether you have installed the **Ansys Fluent Coupling Support** component during Rocky installation, first follow the Install Ansys Coupling Components (p. 792) procedure before beginning Step II.

STEP II: SET UP THE INITIAL CFD SIMULATION

1. In Ansys Fluent, set up the CFD simulation that you want to couple with Rocky. Important:Ensure that you complete all of the following; these exact settings are required for 2-Way Fluent coupled simulations:

   a. From the **Fluent Launcher** dialog, do the following:

      i. Under **Dimension** ensure **3D** is selected.

      ii. Under **Options** select **Double Precision**.

   b. From the **General** tab, under **Time** select **Transient**.

   c. From the **Tree** panel, expand **Models**, and then do one of the following:

      - To set up a CFD project with a single fluid phase (p. 1395), ensure that **Multiphase** model is set to **Off**.

      - Otherwise, to set up a CFD project with two or more fluid phases (p. 1392), do all of the following:

         i. Under **Models**, select **Eulerian**. Note: The **Volume of Fluid** model is not supported by Rocky in this version.

         ii. Under **Eulerian Parameters**, keep all options cleared. (All are unsupported with coupling, including the **Multi-Fluid VOF Model**.)

         iii. Under **Volume Fraction Parameters**, select **Implicit** for **Formulation**.

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

iv. Under **Number of Eulerian Phases**, select the number of phases that includes an additional one for particles. (The number of Eulerian phases should be set to number of fluid phases + 1.).

v. Click **OK**.

d. Only for projects where the **Multiphase** model has been set to **Eulerian** (i.e., multiphase (p. 1392) simulations), ensure that you set the additional particle phase to have a different material from the primary phase.

e. Only for projects involving multiple fluid species, for the **mixture-template** for your **Material Mixture**, ensure that you have enabled **Species Transport**. Important:The species you list last in the **Selected Species** list is very important as Fluent considers it to be the bulk species; this *last species*, therefore, will not result in a mass source term from Rocky.

f. From the **Solution Methods** tab, under **Pressure-Velocity Coupling Scheme**, do one of the following:

- For projects where the **Multiphase** model has been set to **Eulerian** (i.e., multiphase (p. 1392) simulations), you must ensure that the **Phase Coupled SIMPLE** method is selected.
- For projects where the **Multiphase** model has been set to **Off** (i.e., single phase (p. 1395) simulations), you may choose any pressure-velocity coupling scheme that works for your particular application.

g. From the **Solution Methods** tab, under **Transient Formulation**, ensure that **First Order Implicit** is selected.

h. From the **Run Calculation** tab, under **Time Advancement**, do the following:

i. Set **Type** to **Fixed**.

ii. Ensure the **Time Step Size** value you set matches the minimum initial Output Frequency you want for your Rocky and Fluent files. Tip: You can change the outputs to be less frequent than this minimum in your Rocky setup later.

Tip:

To help ensure that you perform these setup steps correctly, please refer to the CFD Coupling Technical Manual document (from the **Help** menu, point to **Manuals**, and then click **CFD Coupling Technical Manual**). For single-phase approaches, you may also follow the self-guided:

Tutorial 14 - DEM-CFD Two-Way Coupling with Ansys Fluent.

Optional Steps:

- If you want to start your Rocky simulation with a given initial fluid flow, save the solution in Fluent as a standard DAT (DAT.H5 or DAT.GZ) file to be used by Rocky later (see Step IV below). Note: If your simulation is multiphase, ensure that the secondary phase (particulate phase) has a volume fraction of zero set before saving your DAT (DAT.H5 or DAT.GZ) file. Later, after the simulation is

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

initialized, the initial particulate volume fraction will be calculated in Rocky according to the particles enabled at the beginning of the simulation.

- If you want to use any standard Fluent post processing options, set them up now before processing the simulation.
- If you want both sides of your coupled simulation to share identical geometries, set up those geometries now in Fluent, but be sure you save your Fluent Case File. Later in Rocky, you can choose to import those geometries directly from the CAS, CAS.H5, or CAS.GZ file that you will save from Fluent in the next step.
- If you want both sides of your coupled simulation to share the same geometry movements, set up the translation or rotation movements now using Fluent Moving Meshes. These movements will later be imported (with the CAS, CAS.H5, or CAS.GZ file) into Rocky (Step IV.2 below), which will automatically convert them into Rocky Motion Frames. Note: If you chose to define your moving meshes in Fluent using expressions, you will need to set up your Motion Frames in Rocky manually. (See also Step IV.2 below and About Creating and Applying Motion Frames to Imported Geometries (p. 533).)

1. Write the completed CFD setup to a CAS (CAS.H5 or CAS.GZ) file.

STEP III: SET UP THE INITIAL ROCKY SIMULATION

1. In Rocky, set up your simulation project as you normally would. (See also Set Simulation Parameters (p. 414).) Tip: If you want to use the exact same geometries as in the Fluent simulation, you may choose to import the CAS, CAS.H5, or CAS.GZ file you generated in Step II, which will include all available geometries as separate components.
2. Recommended: Save your Rocky project to the same folder in which you saved the CAS (CAS.H5 or CAS.GZ) file in Step II. (See also Save a New Simulation Project (p. 853).)

   Whether or not you choose to take this step, Rocky will create its own copy of the CAS (CAS.H5 or CAS.GZ).
3. Optional: If you want to start your coupled simulation with particle data already in the system, do all of the following:

   a. Process the simulation as you normally would, without CFD coupling. (See also Start Processing a Simulation from the Beginning (p. 913).)

   b. After processing is complete, from the **File** menu, click **Save project as…**.

   c. From the **Save As** dialog, choose the third option, "**Save as a New Project for Restart**", and then click **OK**.

   d. In the **Save in** list, choose the same location in which you saved the CAS (CAS.H5 or CAS.GZ) file in Step II, verify the **File Name**, and then click **Save**.

STEP IV: APPLY 2-WAY COUPLING

1. From the **Data** panel, click **CFD Coupling** and then from the **Data Editors** panel, choose **2-Way Fluent** from the **Coupling Mode** list.

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

2. From the **Select Fluent CAS file** dialog, locate and select the CAS (CAS.H5 or CAS.GZ) file you generated in Step II, and then click **Open**. Important:A mesh validation step will occur immediately after the CAS file import. This requires a valid Fluent license on the same machine upon which you are running the Rocky simulation. After successful import, a new **2-Way Fluent** item appears beneath **CFD Coupling** in the **Data** panel. In addition, if you chose to set up Moving Meshes in Fluent, one new Frame will appear under **Motion Frames** in the **Data** panel for each Cell Zone with motions you have defined. Important:If you have defined your Moving Meshes in Fluent using expressions, Rocky will not be able to create the associated Motion Frames for you automatically. Rather, you must manually create your own Motion Frames in Rocky to match the Fluent motions as closely as possible.

3. If the CAS (CAS.H5 or CAS.GZ) file imported new Motion Frames, assign each newly imported Frame to the geometry component you want to share corresponding movements with Fluent. (See also Apply a Motion Frame to an Imported Geometry (p. 569).)

4. From the **Data** panel under **CFD Coupling**, select the new **2-Way Fluent** item and then from the **Data Editors** panel, do all of the following:

   a. From the **Interactions** tab, define the CFD laws you want per *Particle set* (p. 1393) by selecting (or multi-selecting) the set name(s) from the **Particle** list, and then defining the laws you want for the selected set(s). From this tab, you can also set the **Turbulent Dispersion** options for the entire coupled simulation. (See also About Using the 2-Way Fluent Method (p. 795).)

   b. From the **Coupling** tab, define the **Mapping Method** and **Sub-Stepping** options you want.

   c. From the **Zones and Interfaces** tab, do the following:

      - Select one or more options from the **Coupling Fluid Cell Zone** list.
      - (Optional) If provided, select the options you want from the **Mapping Cell Zone Interfaces** list.

   d. From the **Fluent** tab, do all of the following:

      i. **Important:**

      If your Fluent case is multiphase, from the **Rocky Phase** list, ensure that the phase representing particle flow is chosen.

      ii. Choose what fluid flow data Rocky starts with by doing one of the following:

      - To start your Rocky simulation without providing initial fluid flow data, ensure the **Use Data Initialization** checkbox remains cleared. Doing so ensures that the initialization settings prescribed in Fluent will be used to generate the initial fluid flow.
      - To start your Rocky simulation with initial fluid flow data, select **Use Data Initialization**, click the **Import File** button, and then select and open the DAT (DAT.H5 or DAT.GZ) file you saved in Step II. Important:If you chose to use initial fluid flow data in your simulation and your simulation is multiphase (p. 1392), you must have ensured that the secondary phase (particulate phase) had a volume fraction of zero set before you saved your DAT (DAT.H5 or DAT.GZ) file. This is important because later, after the simulation is initialized,

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

the initial particulate volume fraction will be calculated in Rocky according to the particles enabled at the beginning of the simulation.

iii. From the **Version** list, select what Ansys version you want to use for fluid coupling. Tip: If you do not see the Ansys version you expected, see also Rocky does not list my Ansys version when I try to set up 2-Way Fluent Coupling (p. 1326).

iv. Choose how the fluid portion of the simulation is processed by doing one of the following:

- To have it processed on only one processor on your local machine, from the **Execution mode** list, select **Serial**.
- To have it processed on multiple processors on your local machine, from the **Execution mode** list, select **Local Parallel** and then from the **Solver Processes** field, enter the number of processors you want to dedicate to the CFD solver.
- To have it processed across several machines on your network, from the **Execution mode** list, select **Distributed Parallel** and then do one of the following:
  - Under **Hosts**, click the **Add** button for each server you want to add and then for each row, define the **Host name** and **Amount** information.
  - To import an external list of hosts and amounts that you created outside of Rocky, click the **Import File** button, and then from the **Import** dialog, locate and select the TXT file containing the information, and then click **Open**. The **Host** section is populated with the information from the TXT file. Tip: The format for the TXT file needs to include the **Host name** repeated on separate lines for each **Amount** you want devoted. To see an example, create a few sample rows as specified directly above, click **Export File** and then save and view the results as a TXT file.

v. Choose how many Fluent files to keep by doing one of the following:

- To keep each Fluent DAT (DAT.H5 or DAT.GZ) file (and in some cases, as with moving meshes, CAS files as well) for every Rocky output file that is saved, enable the **Keep all files** checkbox. Important:Unless you require all files for post-processing in Fluent, keeping all files is not recommended for the 2-Way Fluent method. (See also About Using the 2-Way Fluent Method (p. 795).)
- To keep only the last (or last several) Fluent DAT (DAT.H5 or DAT.GZ) files (and in some cases, CAS files) saved, ensure the **Keep all files** checkbox is cleared, and then enter the number of last-saved files you want to keep in the **Files to keep** box. Tip: Keeping the last 2 (or more) files is recommended. Important:If you decide to save fewer than all of the files, be aware that you will not be able to post-process the unsaved files in Fluent. Also, be aware that the number DAT (and in some cases, CAS) files you save will be as far back in time as you can restart your simulation since fluid data will not be available for prior times. Tip: You can also choose which quantities and frequency you want to write to disk using Fluent's Autosave functionality.

e. From the **Variables** tab, review the **Additional Inputs** and **Additional Outputs** (if applicable) that Rocky and Fluent will exchange during the coupled simulation. Tip: If you have enabled species transport, this is where the per-species variables Fluent provides Rocky, and the mass source terms Rocky provides Fluent will be listed.

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

Tips:

- To verify or change the CAS (CAS.H5 or CAS.GZ) file selected, click **Open** (at the bottom of the **2-Way Fluent** | **Fluent** tab) and Rocky will open its own copy of the CAS (CAS.H5 or CAS.GZ) file in Fluent. From here, you can view or make changes to the Rocky copy of the file. Important:Because Rocky uses only its own copy of the imported CAS (CAS.H5 or CAS.GZ) file, it is critical that you only open the imported CAS (CAS.H5 or CAS.GZ) from Rocky's **Open** button, and that you save any changes to the default location Rocky indicates.
- If you make changes to the Rocky copy of the CAS (CAS.H5 or CAS.GZ) file in Fluent, click **Refresh** to have Rocky include the updates. Important:The **Refresh** button works only if you first opened the CAS (CAS.H5 or CAS.GZ) file from the Rocky **Open** button located at the bottom of the **2-Way Fluent** | **Fluent** tab.
- If you made changes to your original Fluent version of the CAS (CAS.H5 or CAS.GZ) file and did not use the Rocky **Open** button as specified above, you must set the **CFD Coupling** option to **No Coupling** to clear the values, and then restart "Step IV: Apply 2-Way Coupling" again from the beginning, selecting the updated CAS (CAS.H5 or CAS.GZ) file in step 2.

STEP V: PROCESS THE COUPLED SIMULATION

In Rocky, process the simulation as you normally would (see also Start Processing a Simulation from the Beginning (p. 913)) with the following exception:

- Keeping in mind the **Time Step Size** value you set in Fluent, from the **Solver** | **Time** tab, set what you want for **Fluent Outputs Multiplier**. Tip: You can see how this change affects Rocky's Output Frequency in the disabled **Simulation** field.

Rocky automatically opens Ansys Fluent and the Fluent simulation is processed at the same time as the Rocky simulation, depending upon the limitations of your Ansys license. If any incompatibility is found between the Fluent setup and Rocky, an error will be shown in Rocky with additional information. Once the simulations are complete, the data from both are available to see and analyze however you choose. (See also Analyzing a Simulation (p. 931).)

See Also:

- About Using the 2-Way Fluent Method (p. 795)

2.6.7. Use the 2-Way Fluent Semi-Resolved Method to Process a Fluent and Rocky Simulation

There are five main steps to using the 2-Way Fluent Semi-Resolved method to process a coupled Rocky and Ansys Fluent simulation, as described below.

Note: Both Single phase (p. 1395) and Multiphase (p. 1392) approaches are covered in this procedure.

STEP I: VERIFY THAT ALL ROCKY-FLUENT PROGRAM REQUIREMENTS ARE MET

1. Before beginning Step II below, ensure all of the following program requirements have been met:
    - A Rocky-supported version of Ansys Fluent is installed upon the same machine that contains your Rocky program installation. (See also System Requirements (p. 67).)

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

- If you are not certain whether you have installed the **Ansys Fluent Coupling Support** component during Rocky installation, first follow the Install Ansys Coupling Components (p. 792) procedure before beginning Step II.

STEP II: SET UP THE INITIAL CFD SIMULATION

1. **Important**: When setting up your 3D mesh, ensure that only tetrahedron, prism, pyramid, or hexahedron cell types are used. Note: Polyhedral mesh cells, including the CutCell type, are not supported in this version.

2. In Ansys Fluent, set up the CFD simulation that you want to couple with Rocky. **Important**: Ensure that you complete all of the following; these exact settings are required for 2-Way Fluent Semi-Resolved

   coupled simulations:

   a. From the **Fluent Launcher** dialog, do the following:

      i. Under **Dimension** ensure **3D** is selected.

      ii. Under **Options** select **Double Precision**.

   b. From the **General** tab, under **Time** select **Transient**.

   c. From the **Tree** panel, expand **Models**, and then do one of the following:

      - To set up a CFD project with a single fluid phase (p. 1395), ensure that **Multiphase** model is set to **Off**.

      - Otherwise, to set up a CFD project with two or more fluidphases (p. 1392), do all of the following:

        i. Under **Models**, select **Eulerian**.

        ii. Under **Number of Eulerian Phases**, select the number of phases you need. Note: Unlike with the standard 2-Way Fluent method, you do not need an additional phase for particles with this method.

        iii. Click **OK**.

   d. From the **Solution Methods** tab, under **Pressure-Velocity Coupling Scheme**, you may choose any pressure-velocity coupling scheme that works for your particular application.

   e. From the **Solution Methods** tab, under **Transient Formulation**, ensure that **First Order Implicit** is selected.

   f. From the **Run Calculation** tab, under **Time Advancement**, do the following:

      i. Set **Type** to **Fixed**.

      ii. Ensure the **Time Step Size** value you set is aligned with the **Output Settings** | **Time Interval** value you want to set later in Rocky. Tip: To ensure that your post-processing is better aligned between the two products, it is recommended that the **Time Step Size** in Fluent and the **Output Settings** | **Time Interval** value in Rocky are multiples.

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

Optional Steps:

- If you want to start your Rocky simulation with a given initial fluid flow, save the solution in Fluent as a standard DAT (DAT.H5 or DAT.GZ) file to be used by Rocky later (see Step IV below).
- If you want to use any standard Fluent post processing options, set them up now before processing the simulation.
- If you want both sides of your coupled simulation to share identical geometries, set up those geometries now in Fluent, but be sure you save your Fluent Case File. Later in Rocky, you can choose to import those geometries directly from the CAS, CAS.H5, or CAS.GZ file that you will save from Fluent in the next step.
- If you want both sides of your coupled simulation to share the same geometry movements, set up the translation or rotation movements now using Fluent Moving Meshes. These movements will later be imported (with the CAS, CAS.H5, or CAS.GZ file) into Rocky (Step IV.2 below), which will automatically convert them into Rocky Motion Frames. Note: If you chose to define your moving meshes in Fluent using expressions, you will need to set up your Motion Frames in Rocky manually. (See also Step IV.2 below and About Creating and Applying Motion Frames to Imported Geometries (p. 533).)

3. Write the completed CFD setup to a CAS (CAS.H5 or CAS.GZ) file.

STEP III: SET UP THE INITIAL ROCKY SIMULATION

1. In Rocky, set up your simulation project as you normally would. (See also Set Simulation Parameters (p. 414).)

   Tip:

   If you want to use the exact same geometries as in the Fluent simulation, you may choose to import the CAS, CAS.H5, or CAS.GZ file you generated in Step II, which will include all available geometries as separate components.

   Note:

   2-Way Fluent Semi-Resolved currently supports only the following particle shapes: Spheres, Shells (either rigid or flexible), or Custom Imported Polyhedrons (rigid only).

2. Recommended: Save your Rocky project to the same folder in which you saved the CAS (CAS.H5 or CAS.GZ) file in Step II. (See also Save a New Simulation Project (p. 853).) Whether or not you choose to take this step, Rocky will create its own copy of the CAS (CAS.H5 or CAS.GZ).

3. Optional: If you want to start your coupled simulation with particle data already in the system, do all of the following:

   a. Process the simulation as you normally would, without CFD coupling. (See also Start Processing a Simulation from the Beginning (p. 913).)

   b. After processing is complete, from the **File** menu, click **Save project as…**.

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

c. From the **Save As** dialog, choose the third option, "**Save as a New Project for Restart**", and then click **OK**.

d. In the **Save in** list, choose the same location in which you saved the CAS (CAS.H5 or CAS.GZ) file in Step II, verify the **File Name**, and then click **Save**.

STEP IV: APPLY 2-WAY FLUENT SEMI-RESOLVED COUPLING

1. From the **Data** panel, click **CFD Coupling** and then from the **Data Editors** panel, choose **Fluent Semi-Resolved** from the **Coupling Mode** list.

2. From the **Select Fluent CAS file** dialog, locate and select the CAS (CAS.H5 or CAS.GZ) file you generated in Step II, and then click **Open**. Important:A mesh validation step will occur immediately after the CAS file import. This requires a valid Fluent license on the same machine upon which you are running the Rocky simulation. After successful import, a new **2-Way Fluent Semi-Resolved** item appears beneath **CFD Coupling** in the **Data** panel. In addition, if you chose to set up Moving Meshes in Fluent, one new Frame will appear under **Motion Frames** in the **Data** panel for each Cell Zone with motions you have defined. Important:If you have defined your Moving Meshes in Fluent using expressions, Rocky will not be able to create the associated Motion Frames for you automatically. Rather, you must manually create your own Motion Frames in Rocky to match the Fluent motions as closely as possible.

3. If the CAS (CAS.H5 or CAS.GZ) file imported new Motion Frames, assign each newly imported Frame to the geometry component you want to share corresponding movements with Fluent. (See also Apply a Motion Frame to an Imported Geometry (p. 569).)

4. From the **Data** panel under **CFD Coupling**, select the new **2-Way Fluent Semi-Resolved** item and then from the **Data Editors** panel, do all of the following:

   a. Choose what fluid flow data Rocky starts with by doing one of the following:

      - To start your Rocky simulation without providing initial fluid flow data, ensure the **Use Data Initialization** checkbox remains cleared. Doing so ensures that the initialization settings prescribed in Fluent will be used to generate the initial fluid flow.

      - To start your Rocky simulation with initial fluid flow data, select **Use Data Initialization**, click the **Import File** button, and then select and open the DAT (DAT.H5 or DAT.GZ) file you saved in Step II.

   b. From the **Version** list, select what Ansys version you want to use for fluid coupling. Tip: If you do not see the Ansys version you expected, see also Rocky does not list my Ansys version when I try to set up 2-Way Fluent Coupling (p. 1326).

   c. Choose how the fluid portion of the simulation is processed by doing one of the following:

      - To have it processed on only one processor on your local machine, from the **Execution mode** list, select **Serial**.

      - To have it processed on multiple processors on your local machine, from the **Execution mode** list, select **Local Parallel** and then from the **Solver Processes** field, enter the number of processors you want to dedicate to the CFD solver.

      - To have it processed across several machines on your network, from the **Execution mode** list, select **Distributed Parallel** and then do one of the following:

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

  - Under **Hosts**, click the **Add** button for each server you want to add and then for each row, define the **Host name** and **Amount** information.

  - To import an external list of hosts and amounts that you created outside of Rocky, click the **Import File** button, and then from the **Import** dialog, locate and select the TXT file containing the information, and then click **Open**. The **Host** section is populated with the information from the TXT file. Tip: The format for the TXT file needs to include the **Host name** repeated on separate lines for each **Amount** you want devoted. To see an example, create a few sample rows as specified directly above, click **Export File** and then save and view the results as a TXT file.

d. Choose how many Fluent files to keep by doing one of the following:

  - To keep each Fluent DAT (DAT.H5 or DAT.GZ) file (and in some cases, as with moving meshes, CAS files as well) saved, enable the **Keep all files** checkbox. Important:Unless you require all files for post-processing in Fluent, keeping all files is not recommended for the 2-Way Fluent Semi-Resolved method. (See also About Using the 2-Way Fluent Semi-Resolved Method (p. 831).)

  - To keep only the last (or last several) Fluent DAT (DAT.H5 or DAT.GZ) files (and in some cases, CAS files) saved, ensure the **Keep all files** checkbox is cleared, and then enter the number of last-saved files you want to keep in the **Files to keep** box. Tip: Keeping the last 2 (or more) files is recommended. Important:If you decide to save fewer than all of the files, be aware that you will not be able to post-process the unsaved files in Fluent. Also, be aware that the number of DAT (and in some cases, CAS) files you save will be as far back in time as you can restart your simulation since fluid data will not be available for prior times. Tip: You can also choose which quantities and frequency you want to write to disk using Fluent's Autosave functionality.

Tips:

- To verify or change the CAS (CAS.H5 or CAS.GZ) file selected, click **Open** and Rocky will open its own copy of the CAS (CAS.H5 or CAS.GZ) file in Fluent. From here, you can view or make changes to the Rocky copy of the file. Important:Because Rocky uses only its own copy of the imported CAS (CAS.H5 or CAS.GZ) file, it is critical that you only open the imported CAS (CAS.H5 or CAS.GZ) from Rocky's **Open** button, and that you save any changes to the default location Rocky indicates.

- If you make changes to the Rocky copy of the CAS (CAS.H5 or CAS.GZ) file in Fluent, click **Refresh** to have Rocky include the updates. Important:The **Refresh** button works only if you first opened the CAS (CAS.H5 or CAS.GZ) file from the Rocky **Open** button.

- If you made changes to your original Fluent version of the CAS (CAS.H5 or CAS.GZ) file and did not use the Rocky **Open** button as specified above, you must set the **CFD Coupling** option to **No Coupling** to clear the values, and then restart "Step IV: Apply 2-Way Fluent Semi-Resolved Coupling" again from the beginning, selecting the updated CAS (CAS.H5 or CAS.GZ) file in step 2.

STEP V: PROCESS THE COUPLED SIMULATION

In Rocky, process the simulation as you normally would (see also Start Processing a Simulation from the Beginning (p. 913)) with the following exception:

- Keeping in mind the **Time Step Size** value you set in Fluent, from the **Solver** | **Time** tab, set what you want for **Output Settings** | **Time Interval**. Tip: To ensure that your post-processing is better

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

aligned between the two products, it is recommended that the **Time Step Size** in Fluent and the **Output Settings** | **Time Interval** value in Rocky are multiples.

Rocky automatically opens Ansys Fluent and the Fluent simulation is processed at the same time as the Rocky simulation, depending upon the limitations of your Ansys license. If any incompatibility is found between the Fluent setup and Rocky, an error will be shown in Rocky with additional information. Once the simulations are complete, the data from both are available to see and analyze however you choose. (See also Analyzing a Simulation (p. 931).)

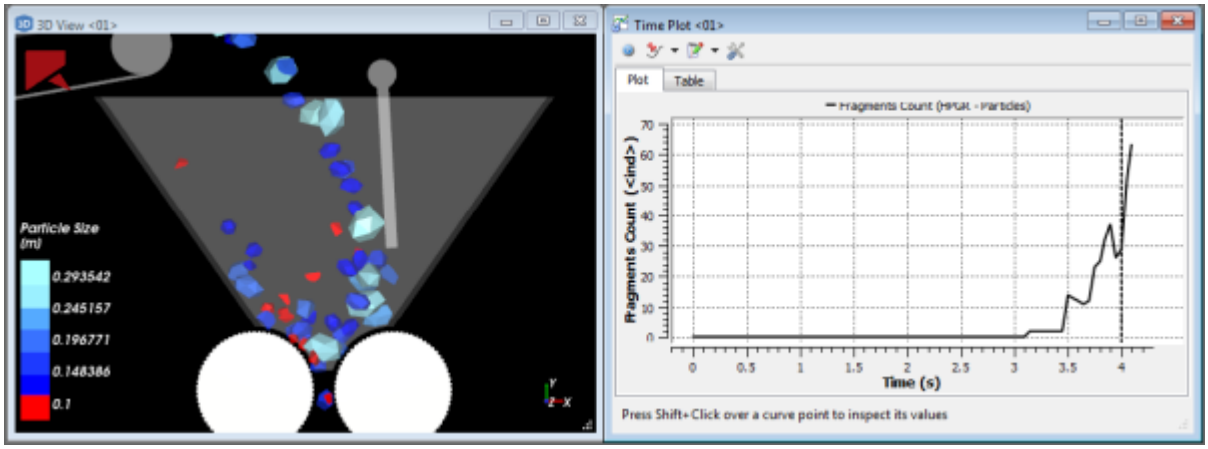
See Also:

- About Using the 2-Way Fluent Semi-Resolved Method (p. 831)

2.6.8. Enable and View Particle Breakage

Analyzing either Instantaneous (p. 1389) or Discrete (p. 1392) particle breakage in Rocky involves first turning on breakage parameters for your *Particle set* prior to processing your simulation (see also About Particle Breakage (p. 665)), and then viewing the results and/or particle-fragment-related Properties or Curves in a 3D View window or graphing them in a plot or histogram window (Figure 1).

Figure 2.176: Example of Instantaneous Breakage being simulated in an HPGR machine



It is important to realize that breakage data is only collected after the Breakage **Start** value on the **Solver** | **Time** tab is reached. This is to allow enough time for the particle flow within your simulation to reach a steady state before breakage calculations begin.

For further details on the specific models, tips, and limitations see the About Particle Breakage (p. 665) topic.

TO ENABLE AND VIEW PARTICLE BREAKAGE:

1. Set up your simulation as you normally would (see also Set Simulation Parameters (p. 414)), ensuring all of the following:

    a. During the Set Simulation-Wide Parameters (p. 414) step, on the **Solver** | **Time** tab, you have set the values you want for both Breakage **Start** and Breakage **Delay after Release**. (See also About Solver Parameters (p. 464).) Tip: It is best practice to set your Breakage **Start** time to begin after a steady state has been reached for your particle flow.

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

b. During the Add and Edit Particle Sets (p. 602) step, you have done all of the following for each of the particle you want breakage enabled:

i. On the **Particle** tab, you have ensured that one of the following **Shape** | **Composition** option combinations are selected:

- Either Straight or Custom *Fiber* Shape | Multiple Element Composition
- Custom *Shell* Shape | Multiple Element Composition
- *Solid* Polyhedron Shape | either Single Element or Multiple Element Composition
- Solid Briquette Shape | Single Element Composition
- Solid Faceted Cylinder Shape | Single Element Composition
- Solid Custom Convex Polyhedron Shape | either Single Element or Multiple Element Composition
- Solid Custom Concave Polyhedron Shape| Multiple Element Composition

ii. If you have selected a **Single Element** particle set, you have done all of the following:

A. On the **Particle** | **Breakage** tab, you have selected the **Enable Breakage** checkbox.

B. From the **Criteria** sub-tab, you have selected the *Instantaneous Breakage* model you want from the **Model** list, and have defined its associated parameters.

C. From the **Fragments** sub-tab, you have entered the values you want under **Limits**, have chosen the model you want from the **Distribution model** list, and have defined its associated parameters.

iii. If you have selected a **Multiple Element** particle set, you have done all of the following:

A. On the **Particle** | **Breakage** tab, you have selected the *Discrete Breakage* model you want from the **Model** list.

B. You have defined the model's associated parameters.

2. Process your simulation as you normally would. (See also About Starting a Simulation (p. 908).)

3. After your simulation reaches as least as far as the Breakage **Start** time, do one or more of the following:

- From a 3D View window (see also About 3D View Windows (p. 1040)), observe the changes to the particle shapes as they fragment and break.
- From the **Data** panel, select **Particles** and then from the **Data Editors** panel, select either the **Properties** or **Curves** tab and then create a plot (see also Graph Data within Rocky by Creating a Plot or Histogram (p. 1124)) or visualize the resulting fragments-related Properties in a 3D View window (see also About 3D View Windows (p. 1040)).

See Also:

- About Particle Breakage (p. 665)

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

- About Collecting Data in Rocky (p. 306)
- About Adding and Editing Particle Sets (p. 602)
- About Properties (p. 933)
- About Curves (p. 989)

2.6.9. Create and View a Particle Assembly

(See also About Particle Assemblies (p. 663).)

TO CREATE AND VIEW A PARTICLE ASSEMBLY:

1. Begin setting up your simulation as you normally would (see also Set Simulation Parameters (p. 414)).
2. During the Add and Edit Particle Sets (p. 602) step, do the following:
    a. For each part (p. 1393) you want represented in your final Assembly (p. 1389) shape, create an individual Particle set (p. 1393). (See also Add a New Particle Set (p. 678).) Tip: Ensure your Particle set meets the specific criteria for Assembly parts. (Refer to the *Assembly Particle Shape Limitations* section of the Particle and Input Limitations (p. 59) topic for details.)
    b. Create another new Particle set (see also Add a New Particle Set (p. 678)), select it, and then from the **Data Editors** panel, do the following:
        i. From the main **Particle** tab, define the **Shape** as an **Assembly**.
        ii. From the **Shape** sub-tab, add a new definition row for each individual part you want to include, and define each's part's position, size scale, and rotation parameters. (See also About Adding and Editing Particle Sets (p. 602).) Tip: Check your settings in the **Particles Details** window. (About Particles Details Windows (p. 679).) Note: For Assembly shapes, the Particles Details 3D view shows the actual position of the part when the size type is Original Size Scale and it is set to 1 (which is the default setting).
    c. Define the rest of the shape settings as you normally would for any other Particle set. (Define what parameters you want on the **Size**, **Movement**, and **Orientation** sub-tabs.) Important:At this point, Rocky will calculate the mass, area, and other properties for the (whole) Assembly particle based on the values you have defined for the Particle sets upon which the individual parts are based, and the size of the parts you have defined within the Assembly. These properties will be calculated accurately by Rocky as long as your individual parts do not overlap; otherwise, the accuracy of these Rocky-calculated properties cannot be guaranteed and you should consider defining them yourself by specifying **Custom Properties**.
    d. (Optional) If your Assembly shape requires its parts to overlap, and you require accurate mass, area, and other properties for your Assembly particle, consider specifying **Custom Properties** for your assembly by doing the following:
        i. From the CAD program with which you designed your ideal (Assembly) particle shape, note the Area, Volume, Mass, Geometric Center, Center of Mass, and other properties you want Rocky to use for your Assembly particle shape.

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

ii. In Rocky, from your Assembly particle's **Particle | Custom Properties** tab, enable the **Change Assembly Properties** checkbox, and then enter the information you want Rocky to use. Tip: You can use the colored dots in the **Particles Details** window to preview where your Center of Mass (blue dot) and Geometric Center (yellow dot) are located.

3. Finish setting up your simulation, process it, and then analyze the results as you normally would.

See Also:

- About Particle Assemblies (p. 663)
- About Adding and Editing Particle Sets (p. 602)
- Particle and Input Limitations (p. 59)
- About Defining and Importing Custom Particle Shapes (p. 670)

2.6.10. Enable Thermal Modeling Calculations

If you want to observe how particles are affected by temperature changes brought on by the heating or cooling of other items around it, you may set up your simulation to model thermal properties. Turning on Thermal Modeling enables you to simulate conductive heat transfer from particles to other particles, and from particles to boundaries. When used with CFD Coupling methods, it can also simulate convective heat transfer between particles and fluids. (See also Set or Modify Fluid and/or Air Flow Properties (p. 757).)

Tip:

For examples and walk-through tutorials of thermal modeling in action, review the following resources:

- Tutorial - Conical Dryer in the *Rocky Tutorial Guide*
- Tutorial - Windshifter in the *Rocky Tutorial Guide*

TO ENABLE THERMAL MODELING IN DEM CALCULATIONS:

1. Set up your simulation as you normally would (see also Set Simulation Parameters (p. 414)), ensuring all of the following:

   a. During the Set Simulation-Wide Parameters (p. 414) step, on the **Physics** tab, select the **Enable Thermal** checkbox. (See also About Physics Parameters (p. 417).) Note: If you decide to enable Intra-particle Collision Statistics, know that the Temperature property will not be available to view on the **Particles Details** window, but can still be viewed from the main **Particles** entity or through a **3D View** window. (Se also About Intra-Particle Collision Statistics (p. 324).)

   b. During the Add and Edit Geometry Components (p. 480) step, on the **Geometry** sub-tab for each of your wall components, set what you want for **Thermal Boundary Type** and **Temperature**.

   c. During the Add and Edit Particle Sets (p. 602) step, on the **Particle** tab for the Particle set(s) you want affected by Thermal, ensure that on the **Composition** sub-tab for any particles you

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

have you have selected **Multiple Elements** from the **Composition** list, that you also set the **Conductivity Ratio** parameter to the value you want.

d. During the Modify Material Properties (p. 591) step, on the **Materials** tab for each of your particle and/or boundary materials, set all of the following parameters:

- **Thermal Conductivity**
- **Specific Heat**
- **Poisson's ratio** (See also About Modifying Material Compositions (p. 591).)

e. During the Add and Edit Particle Inputs (p. 708) step, for each Particle Input, set the **Temperature** of each Particle set row. (See also About Adding and Editing Particle Inputs (p. 708).)

f. If coupling your simulation with CFD methods, then during the Set or Modify Fluid and/or Air Flow Properties (p. 757) step, set what you want for the **Convective Heat Transfer Law**. (See also About Using the 1-Way Fluent Method (p. 775), About Using the 2-Way Fluent Method (p. 795), or About Using the 1-Way Constant Method (p. 763)). Note: If coupling your simulation with the 1-Way Constant method specifically, ensure you also set the values you want for constant **Temperature**, **Thermal Conductivity**, and **Specific Heat**. (See also About Using the 1-Way Constant Method (p. 763).)

2. Process your simulation as you normally would. (See also About Starting a Simulation (p. 908).)

TO ENABLE THERMAL MODELING IN SPH CALCULATIONS:

1. Set up your simulation as you normally would (see also About SPH Parameters (p. 436)), ensuring all of the following:

a. During the Set Simulation-Wide Parameters (p. 414) step, on the **Physics** tab, select the **Enable Thermal** checkbox. (See also About Physics Parameters (p. 417).) The Cleary model for thermal transfer is automatically selected.

b. During the Add and Edit Geometry Components (p. 480) step, on the **Geometry** sub-tab for each of your wall components, set what you want for **Thermal Boundary Type** and **Temperature**.

c. **If you simulation also involves DEM particles**: During the Add and Edit Particle Sets (p. 602) step, on the **Particle** tab for the Particle set(s) you want affected by Thermal, ensure that on the **Composition** sub-tab for any particles you have you have selected **Multiple Elements** from the **Composition** list, that you also set the **Conductivity Ratio** parameter to the value you want.

d. During the Modify Material Properties (p. 591) step, on the **Materials** tab set all of the following parameters for the fluid material:

- **Thermal Conductivity**
- **Specific Heat**

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

2. Process your simulation as you normally would. (See also About Starting a Simulation (p. 908).)

Tip:

When you are ready to analyze your simulation (see also Analyzing a Simulation (p. 931)), you might pay special attention to the following Properties (see also About Properties (p. 933)):

- Temperature (for walls, conveyors, particles and fluids)
- Specific Heat (only for fluids)
- Thermal Conductivity (only for fluids)

See Also:

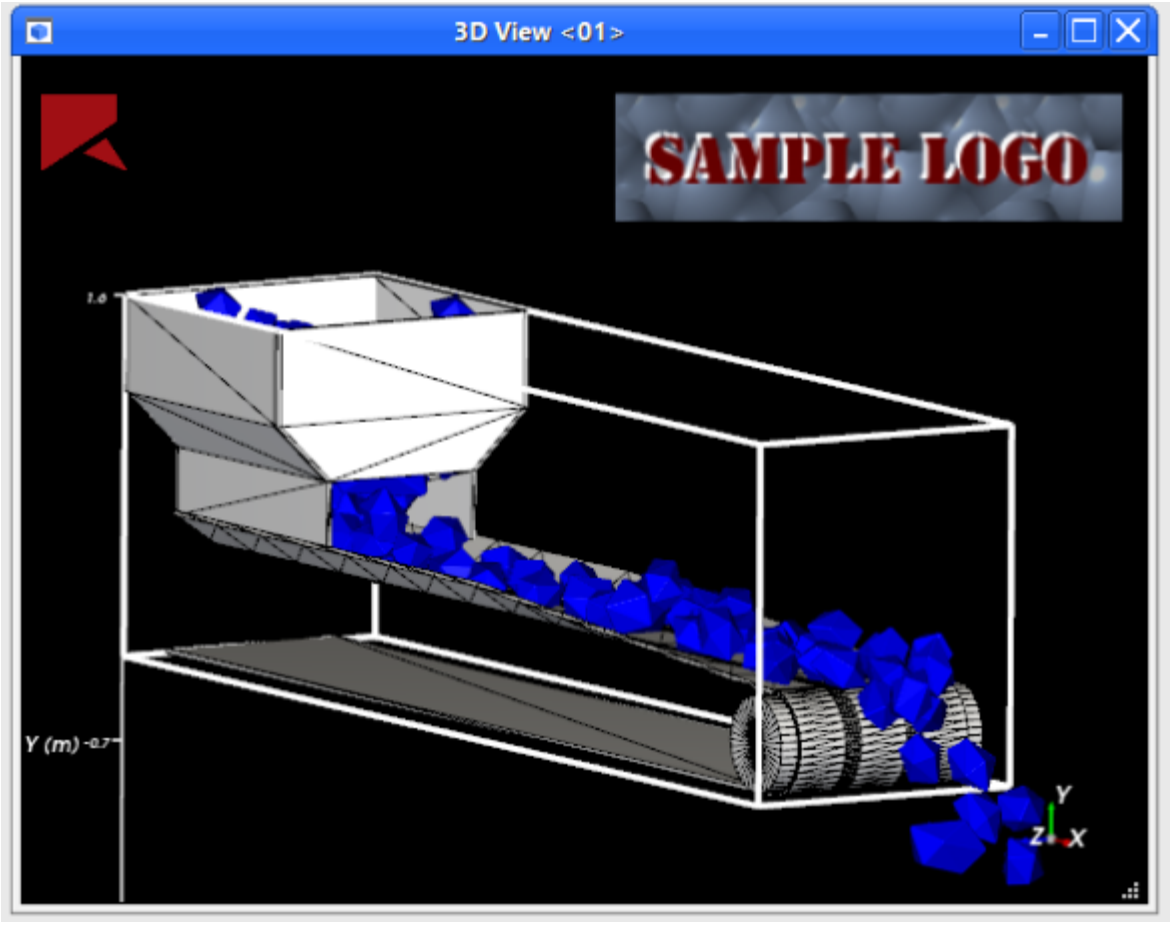
- View Frequently Used Tasks (p. 371)

2.6.11. Add a Logo to a 3D View

A 3D View of your simulation will always contain the Rocky logo but you can choose to add your own company logo to the view, too. This may be useful in cases where you are showing simulation images or animations to clients and want to brand your work.

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

Figure 2.177: 3D View window with a Sample Logo added as an Image Overlay



You add a logo image by using the Overlay tab on the **Window Editors** panel. Logos are added as images and can be positioned anywhere on the 3D View window that you like.

TO ADD A LOGO TO A 3D VIEW:

1. Using separate image editing software, open the logo file you want to use and ensure the display size is appropriate.

2. Ensure your image is saved as either a PNG, JPG, BMP, or PNP file.

3. Open the simulation project to which you want to add the logo and select the 3D View window you want.

4. Using the **Overlay** tab on the **Window Editors** panel, add the logo to the 3D View as an Image Overlay. (See also Add or Edit an Image Overlay for a 3D View Window (p. 1070).)

See Also:

- About Using the Window Editors panel to Change a 3D View (p. 1058)
- View Frequently Used Tasks (p. 371)

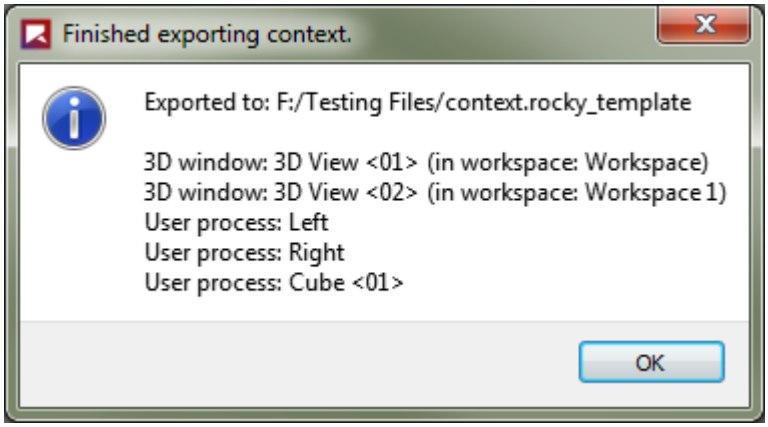
Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

2.6.12. Reuse 3D View Window, Workspace, and User Process Setups in Other Projects

For projects where you have spent time and effort setting up several 3D View windows, Workspaces (p. 1396), and User Processes just the way you like, you can now export all of that setup criteria and reuse it in other projects. This is an alternative to recording or writing a script (p. 1394) to do the setup for you, which requires you to know beforehand exactly how you want your views and processes to work. With this feature, you can decide after the fact whether you want to reuse the setup for another project or not.

This can be useful in cases where you have already run several similar simulations but your client suddenly wants to see new views or processes. By using this feature, you can set up the new views and processes in one project, and then apply those same views and processes to the rest of the projects in your set.

Rocky enables you to export this setup criteria to either a `rocky_template` file or to the clipboard, and even lists each component that was exported in a separate dialog (Figure 1) for confirmation purposes.



Example dialog that appears after exporting project context

In your new project, you can then choose to import the saved criteria from those same two locations.

Once imported, Rocky simply creates new 3D View windows, Workspaces, and User Processes to match what was exported. If you already have 3D View windows and User Processes in your project at the time of Import, Rocky will retain these items while adding the new ones to the project.

TO EXPORT 3D VIEW WINDOW, WORKSPACE, AND USER PROCESS SETUPS FOR USE IN OTHER PROJECTS:

1. Ensure that your 3D View windows, Workspaces, and User Processes are set up the way you want.

2. From the **File** menu, point to **Export project context** and then do one of the following:

- To ensure reuse of the project context for longer than just the immediate timeframe, select **To file**, and then do all of the following: a. From the **Save project context** dialog, select the location to which you want to save the `rocky_template` file, name it in the **File name** box, and then click **Save**. b. In the **Finished exporting context** dialog, review the items listed, and then click **OK**.

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

- To reuse the project context for only the immediate timeframe, select **To clipboard**, and then in the **Finished exporting context** dialog, review the items listed, and then click **OK**.

  The 3D Window, Workspace, and User Process setups are ready to import into another project.

TO IMPORT 3D VIEW WINDOW, WORKSPACE, AND USER PROCESS SETUPS PREVIOUSLY EXPORTED OUT OF ANOTHER PROJECT

1. Ensure the project that you want to apply the previously exported setups is open in Rocky.
2. From the **File** menu, point to **Import project context** and then do one of the following:

- To import from a previously exported file, select **From file**, and then from the **Restore project context** dialog, locate and select the `rocky_template` file you want to use, and then click **Open**.
- To import from the clipboard, select **From clipboard**.

  The 3D Window, Workspace, and User Process setups you previously exported are added to the project.

See Also:

- About Creating and Using PrePost Scripts (p. 296)
- Create and Modify a 3D View (p. 1040)
- Filter Views and Data with User Processes (p. 1174)
- About Workspaces (p. 108)
- View Frequently Used Tasks (p. 371)

2.6.13. Set Up a New GPU Card

When setting up a new GPU card for use with Rocky, there are two steps you can take that might help make processing your simulations on GPU smoother and faster:

- Turn off the Windows Timeout Detection and Recovery (TDR)
- Enable double-precision processing

See below for specific instructions.

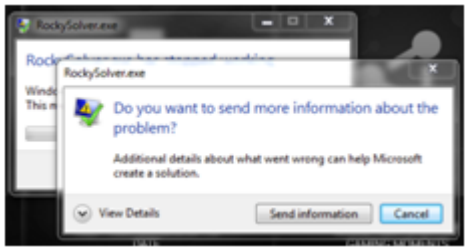
TURN OFF TIMEOUT DETECTION AND RECOVERY (TDR) FOR A GPU CARD

Application: This procedure is particularly useful when you have a single NVIDIA GPU card. But it can also be useful for any GPUs used by Rocky in general.

Symptom: Your Rocky simulations sporadically stop during the middle of a complex simulation, resulting in an incomplete calculation and an error (Figure 1). This is caused by Windows believing the GPU card is non-responsive because it is taking longer than expected to return an answer on a difficult calculation. Turning off TDR for your GPU allows the card the necessary time to execute the computation.

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

Figure 2.178: Error seen in Rocky when TDR causes the GPU to stop processing



To turn off TDR for a GPU card:

1. Ensure that you have successfully installed your GPU card and associated drivers, and have restarted your computer.

2. Download and install the latest NVIDIA Nsight Visual Studio Edition from Nvidia.

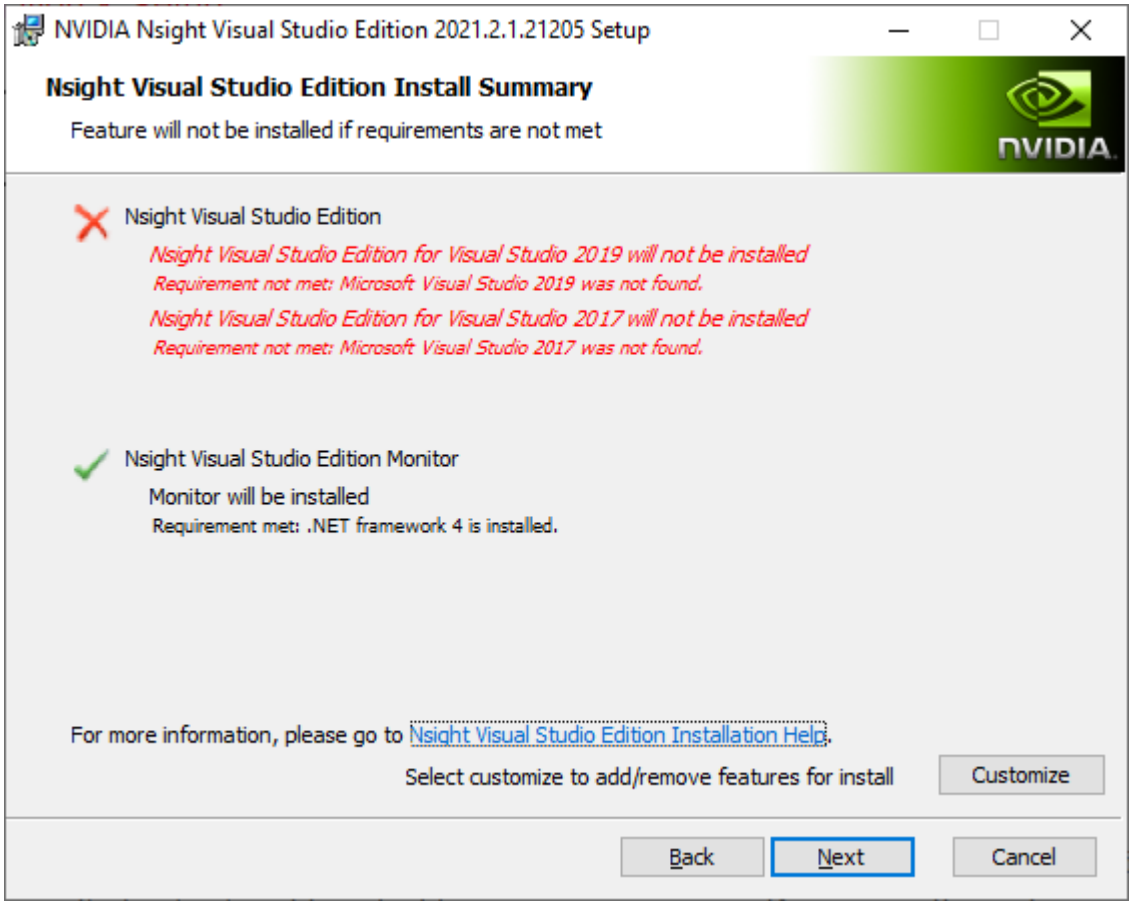


Figure 2: Installation confirmation screen for Nsight Visual Studio Edition

Notes:

- As seen in Figure 2 above, you will need only to install the Nsight Monitor option; the Nsight Visual Studio option is not necessary.

- You may need to Subscribe as a developer (free of charge) on the Nvidia website, if you haven't done it before, in order to download the Nsight Visual Studio Edition (Figure 3).

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

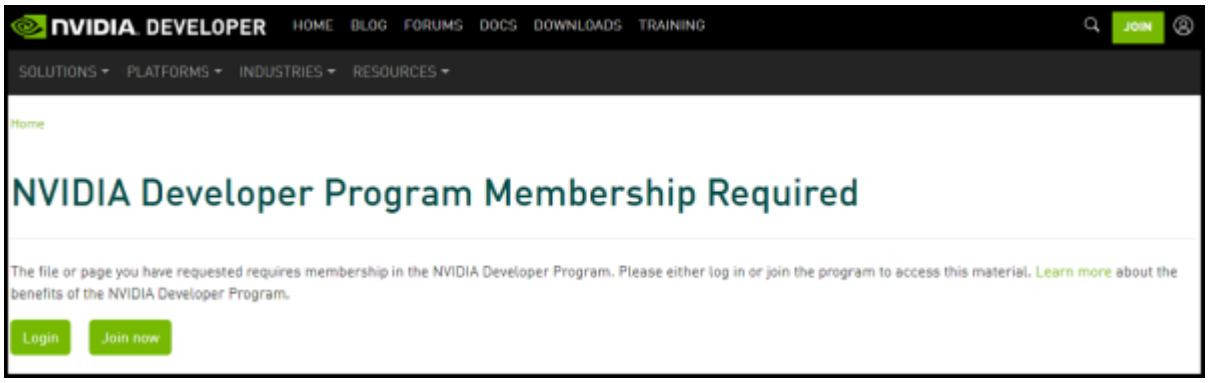


Figure 3: Subscribe to Nvidia developer program

3. Restart your computer.
4. From the Windows **Start** menu, under **NVIDIA Corporation**, click **Nsight Monitor** (Figure 4).

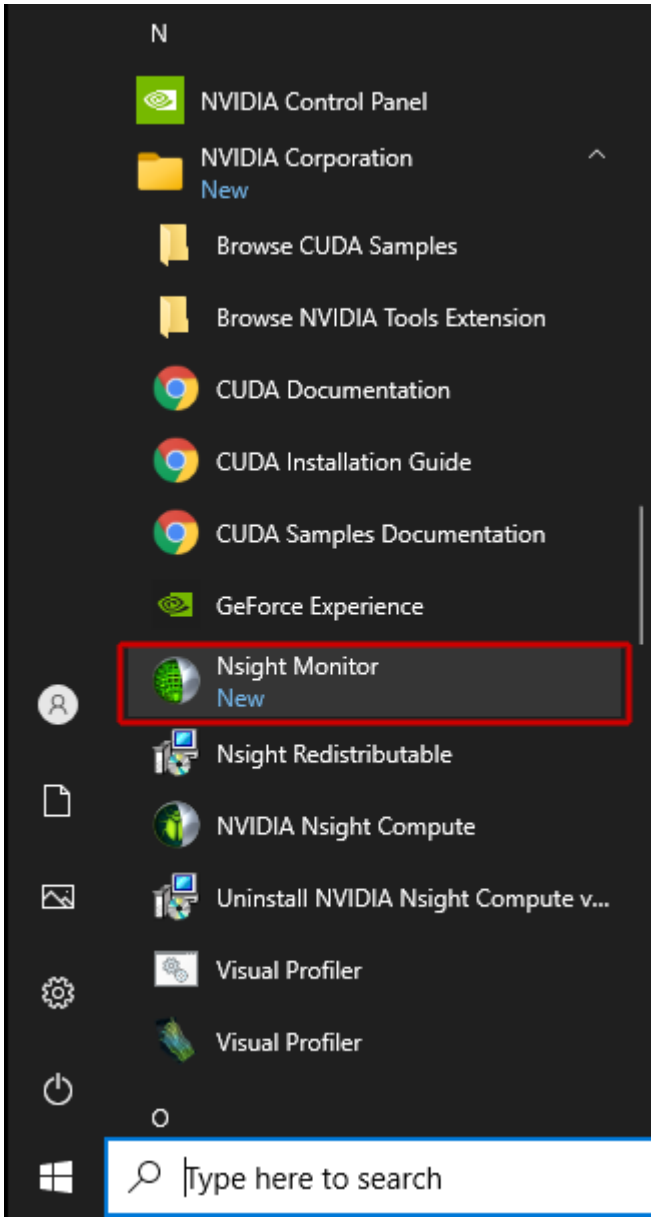


Figure 4 Nsight Monitor in Windows Start menu A new **Nsight Monitor** icon appears in your Notification Area.

5. From your Notification Area, right-click the **Nsight Monitor** icon and then select **Options** (Figure 5).

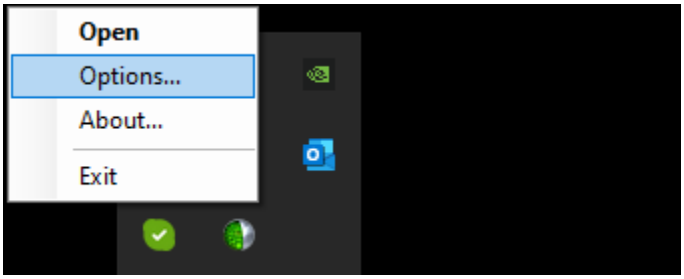


Figure 5: Nsight Monitor in Notification Area

6. From the **NVIDIA Nsight Options** dialog, on the **General** tab, set **WDDM TDR enabled** to **False** (Figure 6).

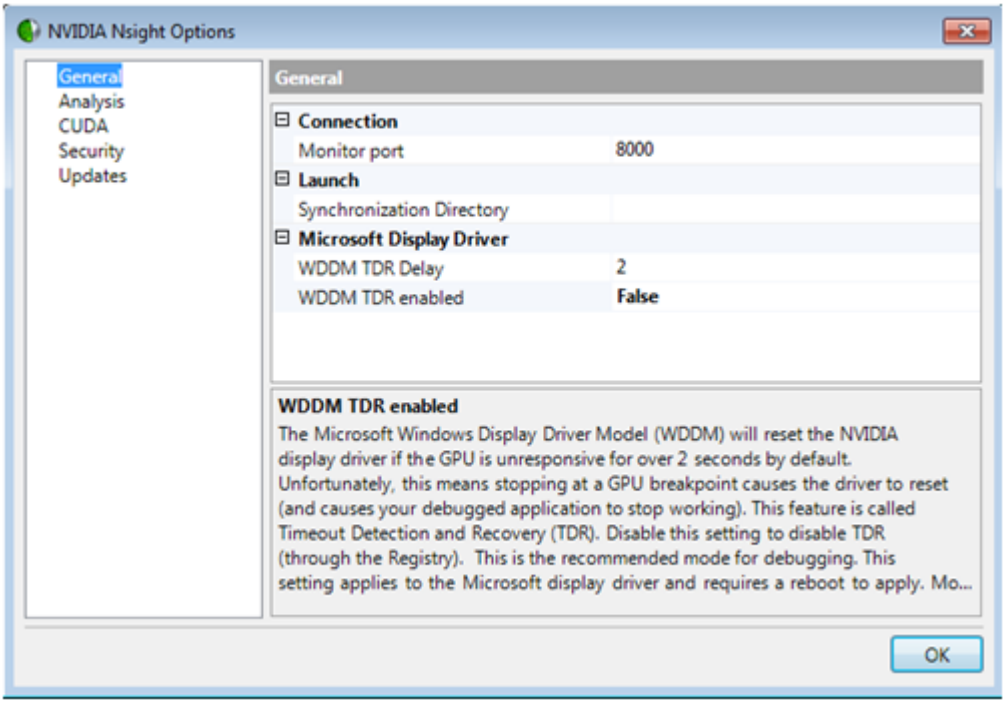


Figure 6: NVIDIA Nsight Options dialog

7. Click **OK**.

8. Restart your computer.

ENABLE DOUBLE-PRECISION PROCESSING FOR YOUR GPU CARDS

Application: You have one or more GPU cards from the NVIDIA Titan family.

Symptom: You are seeing slower than expected results on your Titan family GPU card. This could be a result of not yet enabling double-precision processing. Doing so will speed up certain Rocky calculations, thereby reducing your overall processing time, at the expense of slowing down your graphics visualization performance. Additionally, it has been found that a Titan card using double precision will take longer when simulating Spheres (~20%), but will be faster when simulating shaped particles (p. 1395) (~200%).

To enable double-precision processing for your GPU cards:

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

1. Ensure that you have successfully installed your Titan card(s) and associated drivers, and have re-started your computer.

2. Right-click your Windows desktop, and then click **NVIDIA Control Panel** (Figure 7).

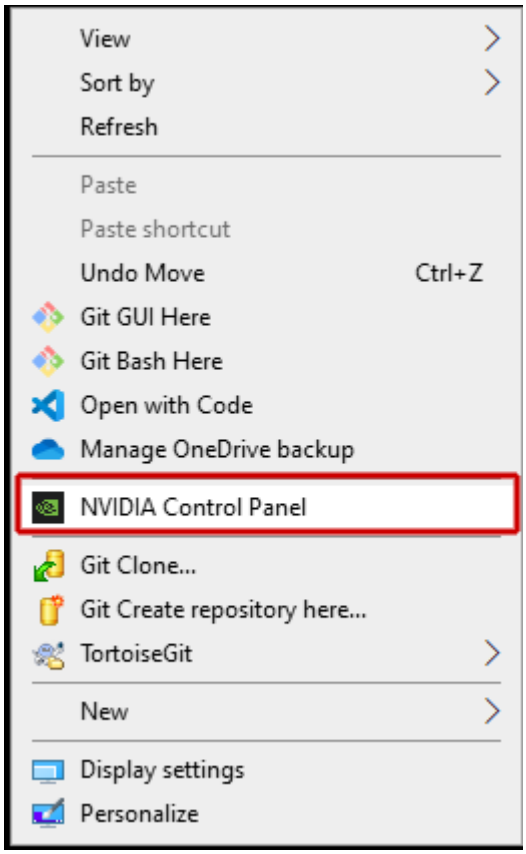


Figure 7: Windows desktop right-click menu showing NVIDIA Control Panel option

3. In the **NVIDIA Control Panel** window, do all of the following:

   a. From the **Select a Task** list, select **Manage 3D settings**.

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

b. In the **Global Settings** tab under **Feature**, select **Double precision** and then from the drop-down list, ensure all your Titan GPU cards are enabled (checked) (Figure 8).

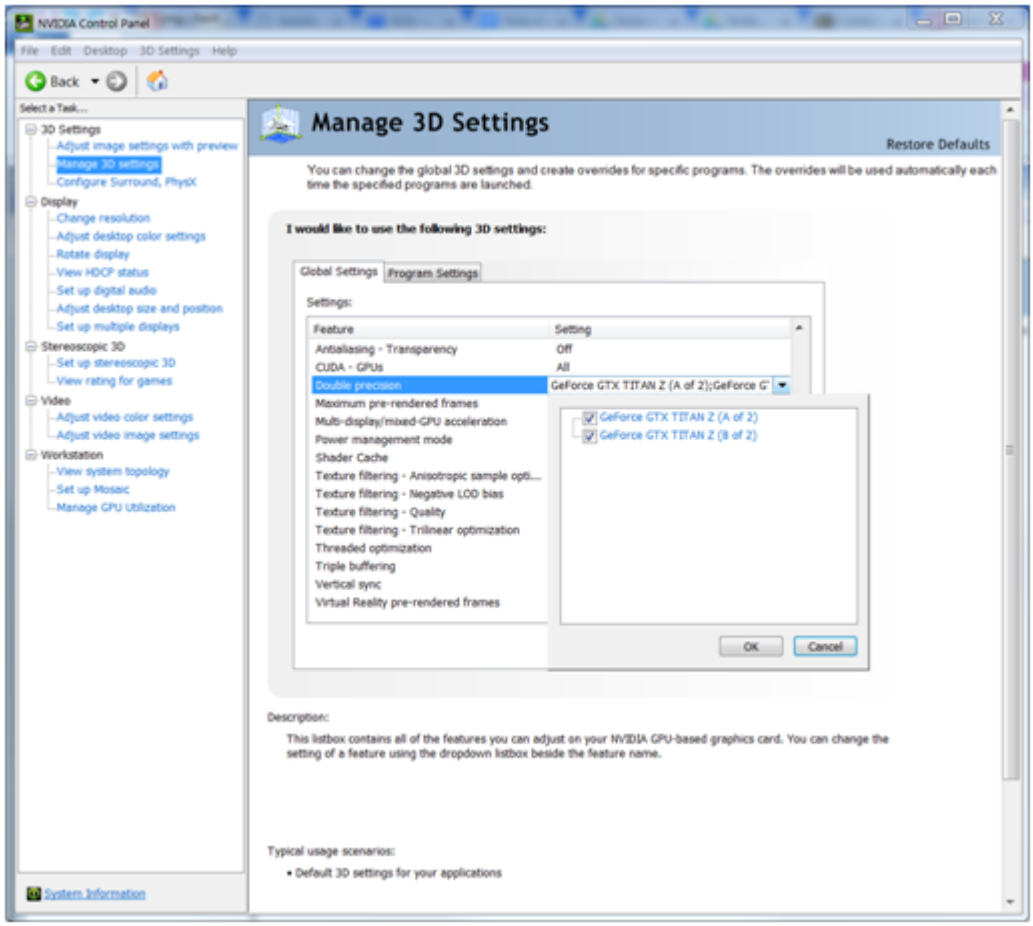


Figure 8: Manage 3D Settings section in the NVIDIA Control Panel

c. Click **OK**.

4. Restart your computer.

See Also:

- Best Practices for Faster Processing (p. 68)

2.7. PrePost Scripting to customize Ansys Rocky software

Recording or writing PrePost Scripts (p. 1394) and then playing or using them in Rocky are great ways to save time by automating repeatable tasks.

Python is the programming language upon which Rocky is built. While knowing how to understand Python is not a requirement for using Rocky, there are several ways you can use the language to customize and extend the features included within the program.

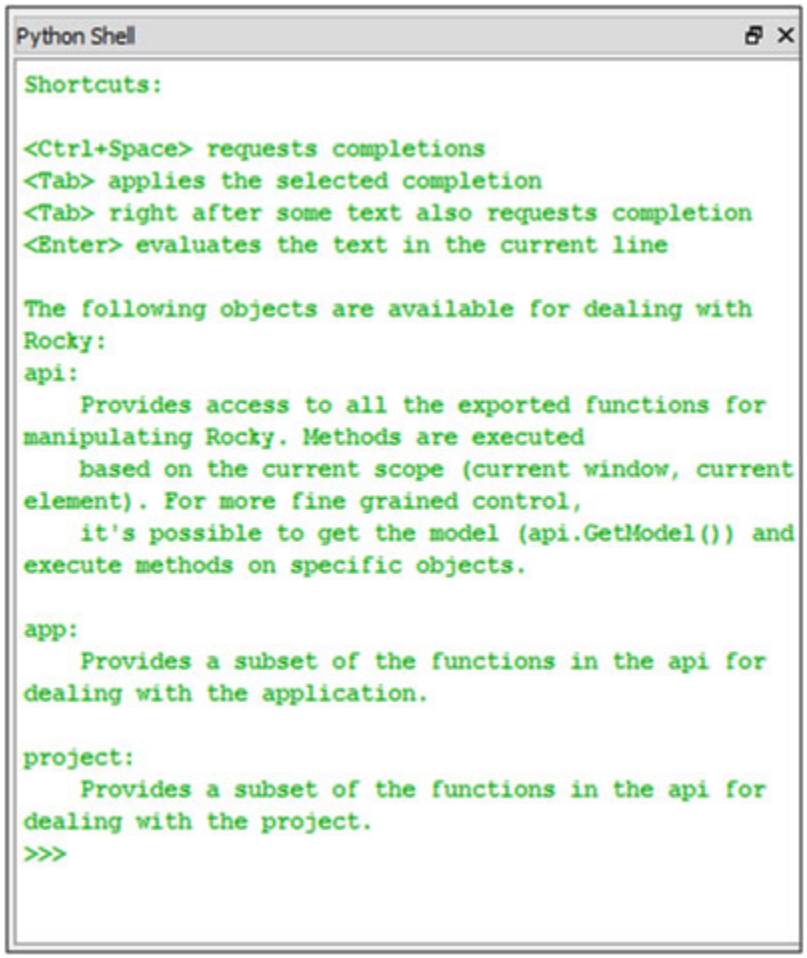
For example, one way in which Python is useful is for creating and editing scripts (p. 1394).

You can also use Python to customize an Advanced Expression for a Property or Curve function. (See also About Customizing Properties or Curves (p. 1012).)

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

Rocky also includes a special panel that you can use to edit the underlying Python code and explore Rocky's PrePost Scripting functionality. To open this panel, from the **Tools** menu, click **Python Shell**.

Figure 2.179: Python Shell panel



Note: This version of Rocky supports only the Python 3 language. If your scripts were built using an older version of Rocky-or an older version of Python (for example, Python 2.7)-they might no longer work in this version. (See also My old script no longer works (p. 1375).)

Tips:

- Find Rocky APIs quickly by using the **Contents**, **Search**, or **Index** tabs in the Rocky PrePost Scripting Manual. (From the Rocky **Help** menu, point to **Manuals** and then click **PrePost Scripting Manual**.)
- View and modify sample scripts in the PrePost Scripts Repository section on the Rocky Customer Portal. (If you do not already have sign-in information for the customer portal, please contact your Rocky representative for assistance.)

See Also:

- I cannot find Rocky API documentation (p. 1317)
- Set Your Rocky Preferences (p. 226)
- Learn About Creating and Using PrePost Scripts (p. 296)

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

- Record a New Script (p. 301)
- Add an External Script File to the Scripts Directory (p. 301)
- Manually Apply a Saved Script (p. 302)
- Automatically Apply Scripts Directly Before or After Processing (p. 303)
- Edit a Saved Script Using the Python Programming Language (p. 303)
- Copy a Script (p. 304)
- Remove a Script (p. 305)
- Rename a Script (p. 305)

See Also:

- Set Your Rocky Preferences (p. 226)

2.7.1. About Creating and Using PrePost Scripts

Creating and using PrePost Scripts (p. 1394) in Rocky is a great way to save time by automating repeatable tasks. Scripts can be recorded and played in the **PrePost Script** panel, added to the **PrePost Scripts** list under the **Solver** entity, and can be launched from the command line (see also Use the Command Line to Play a Script (p. 1313)).

How you choose to create a script depends upon the task you are trying to repeat, and your level of familiarity with both Python and Rocky's PrePost Scripting functionality (see also Customize and Extend Rocky with Python (p. 294)). Specifically:

- If the set of steps you want to repeat can be limited to things you click or enter in the Rocky UI, recording a script in the **PrePost Script** panel might be your easiest and quickest choice.
- If you require more control over inputs or want access to tasks not directly accessible from the Rocky UI, then writing a script in Python may work better for your needs.

See the sections below for more detail.

ABOUT RECORDING AND PLAYING SCRIPTS IN THE PREPOST SCRIPT PANEL

When you choose to start recording a script from the **PrePost Script** panel, every step you take in Rocky is remembered until you choose to stop the recording. Then, you can choose to "play" your script within other parts of the set up, another view, or even other simulation files to repeat the same recorded steps automatically.

For example, you can use recorded scripts to accomplish all of the following tasks:

- Create the very same Animation Key Frame sets in other simulations
- Create a common variable set for use in other simulations
- Apply custom graph views to your plot window that use your preferred units and grid settings

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

- Export a set of animation key frames to individual image files
- Apply your preferred visualization settings to other 3D View windows within your same simulation

When you record a script using the **PrePost Script** panel, Rocky automatically saves the resulting PY file in the correct folder location and using the required *script_* prefix.

Tip: As long as your script PY file begins with *script_* and is saved in either the *..RockyScripts* folder (for scripts shared across projects) or the *..<ProjectName>rocky.filestmpmacros-0* folder (for current-project-only scripts) (see also File Types and Folders in Rocky (p. 129)), it can be listed and played in the **PrePost Script** panel.

ABOUT WRITING PREPOST SCRIPTS IN PYTHON

You can also write and edit scripts outside of the Rocky program, and then apply them to your simulations in Rocky-either by playing the resulting PY file from the **PrePost Script** panel, or by pasting the code directly into the **Python Shell** panel.

To write or edit a script, knowledge of both Python (see also Customize and Extend Rocky with Python (p. 294)) and Rocky's PrePost Scripting functionality (see also I cannot find Rocky API documentation (p. 1317)) is required.

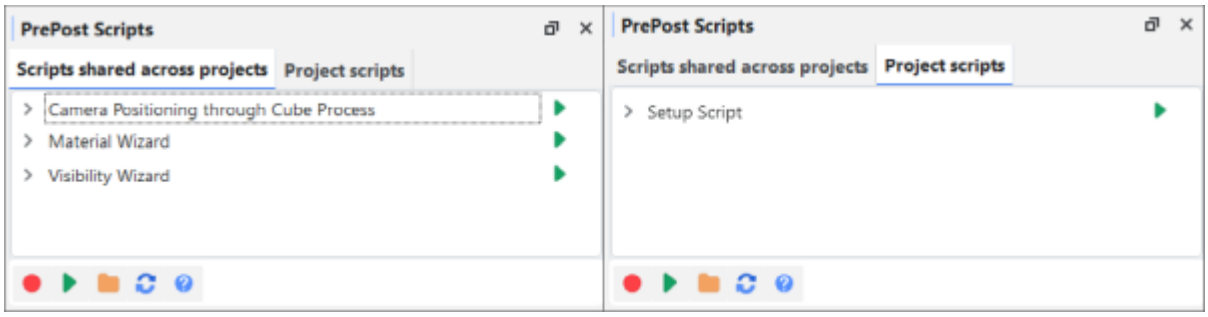
Tips:

- Gain hands-on experience with creating and using scripts by following either Tutorial - Static Angle of Repose Test in the *Rocky Tutorial Guide* or Tutorial - Tablet Coater in the *Rocky Tutorial Guide*.
- View and modify sample scripts in the PrePost Scripts Repository section on the Rocky Customer Portal. (If you do not already have sign-in information for the customer portal, please contact your Rocky representative for assistance.)

ABOUT SHARED AND PROJECT-ONLY SCRIPTS

Rocky enables you to share your recorded and saved scripts across different Rocky projects, or use them for only the current project. Scripts saved to the **Scripts shared across projects** tab will be available for all your Rocky projects across the application; scripts saved to the **Project scripts** tab will appear only in the current project (Figure 1).

Figure 2.180: PrePost Script panel showing different scripts on both tabs



The tab you have selected when you choose to record your script (see also Record a New Script (p. 301)) or import your script (see also Add an External Script File to the Scripts Directory (p. 301)) determines

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

the category to which the new script will be assigned. Scripts can be copied from one tab to another through the right-click menu (see also Copy a Script (p. 304)).

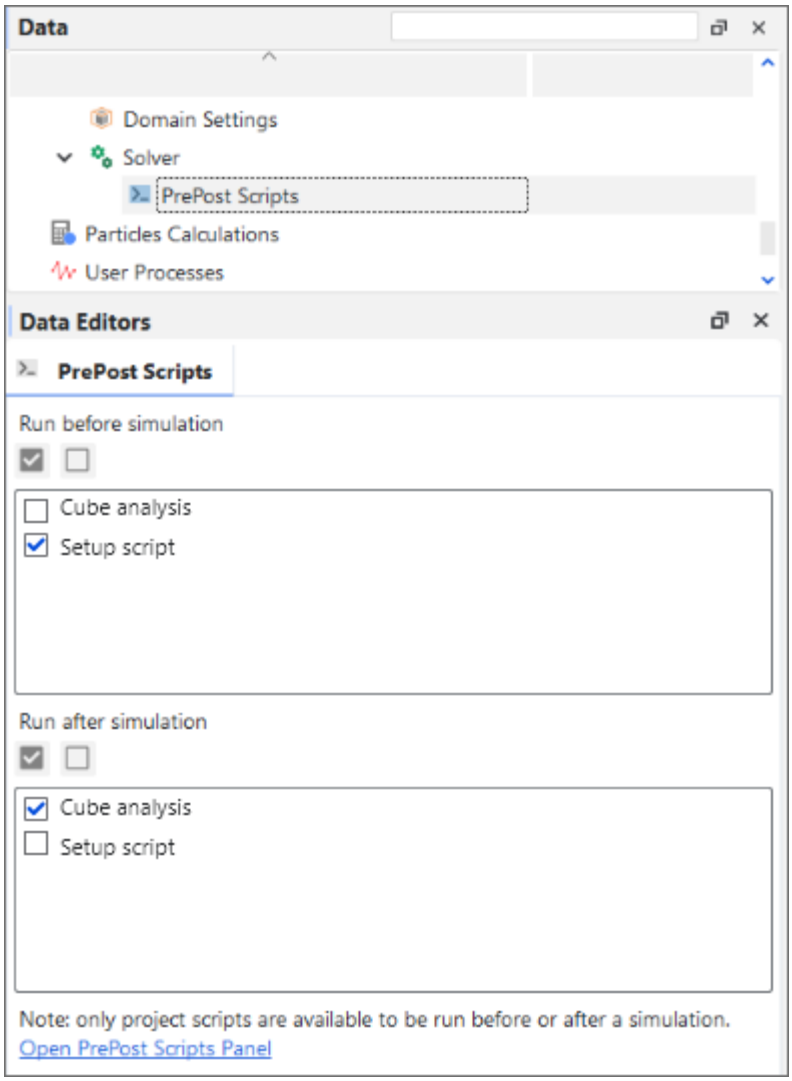
Note:

Only scripts on the **Project scripts** tab are saved during archiving (see also Archive a Simulation Project (p. 1294)) and are shown on the **PrePost Scripts** entity under **Solver** (see below.)

ABOUT SOLVER, PREPOST SCRIPTS

Any scripts that you have saved to the **Project scripts** tab are eligible for running automatically before and/or after simulation processing through the **PrePost Scripts** sub-entity on the **Data** panel under **Solver** (Figure 2). In this way, you can further automate your setup and post-processing, especially in situations where you are running many similar kinds of cases.

Figure 2.181: Solver, PrePost Scripts sub-entity in Data Editors panel



ABOUT SOLVER, PREPOST SCRIPTS

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

In this version of Rocky, only scripts that are compatible with the Python 3 (PY3) language will be supported. This means that some scripts that worked in older versions of Rocky may need to be re-created or modified to support the Python 3 guidelines. (See also My old script no longer works (p. 1375).)

In addition, only PY files starting with *script_* will be eligible to play in the **PrePost Script** panel. Script files without this prefix can still be applied by copying and pasting the code into the **Python Shell** panel.

PREPOST SCRIPT SETTINGS

Use the images and table below to understand how to create and apply scripts.

Figure 2.182: PrePost Script panel showing no scripts shared across projects

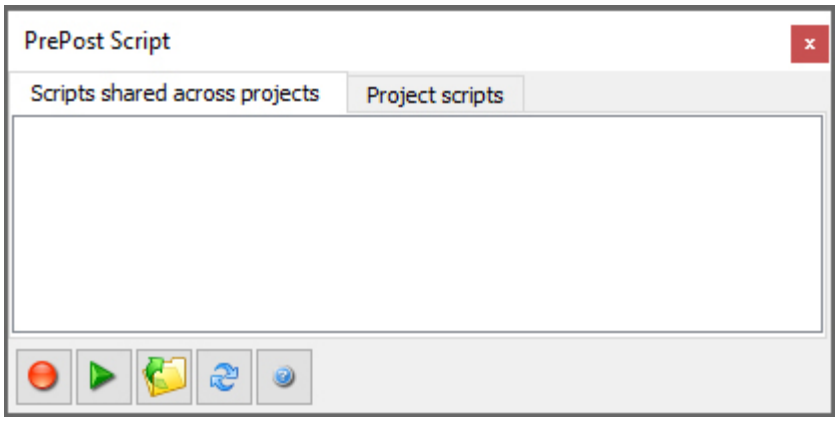


Figure 2.183: New PrePost Script dialog

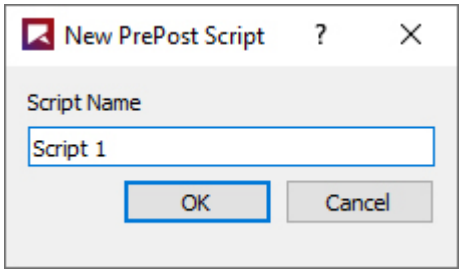


Table 1: PrePost Script panel button descriptions

Button	Description
Record Script	Starts the recording process, which is the first step to creating a new script. Saves each step in Rocky until you press the **Stop and Save Script** button. Note: The tab you have selected when you click **Record Script** determines what kind of script is created-a script shared across all projects, or a script saved only for this project.
Stop and Save PrePost Script	Stops the recording process and saves the steps that were taken.

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

Button	Description
 Playback PrePost Script	Applies the recorded steps of the selected script to whatever element is currently active in the Rocky UI.
 Open PrePost Scripts Directory	Opens the tab-specific (shared across project or just current project) Scripts folder where the script PY files are saved. From here, you can choose to do any of the following: • Modify your saved script files by editing them in the Python programming language. • Send your script to another Rocky user. • Add a new script to the directory. Note: The tab you have selected when you click **Open PrePost Scripts Directory** determines which scripts folder is opened-the folder containing scripts that are shared across all projects, or the folder containing scripts saved only for this project.
 Reload PrePost Scripts From the Filesystem	When scripts are added to or removed from the directory, this refreshes the list in the **PrePost Script** panel to include any newly added or removed PY files.
 Help	Opens a short Help file with instructions for how to use the **PrePost Script** panel.

What do you want to do?

- Record a New Script (p. 301)
- Add an External Script File to the Scripts Directory (p. 301)
- Manually Apply a Saved Script (p. 302)
- Automatically Apply Scripts Directly Before or After Processing (p. 303)
- Edit a Saved Script Using the Python Programming Language (p. 303)
- Copy a Script (p. 304)
- Remove a Script (p. 305)
- Rename a Script (p. 305)
- Use the Command Line to Play a Script (p. 1313)

See Also:

- Set Your Rocky Preferences (p. 226)
- Reuse 3D View Window, Workspace, and User Process Setups in Other Projects (p. 403)
- My old script no longer works (p. 1375)

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

2.7.2. Record a New Script

1. Ensure the **PrePost Script** panel is displayed. (From the **Tools** menu, click **PrePost Script**.)
2. From the **PrePost Script** panel, do one of the following: - To record a script that can be shared across all other Rocky projects, ensure the **Scripts shared across projects** tab is selected. - To record a script that can be used only for this project, ensure the **Project scripts** tab is selected.

   Tip:

   If you change your mind, you can copy the script to the other tab later. (See also Copy a Script (p. 304).)

3. Click the **Record Script** button.
4. From the **New PrePost Script** dialog, enter a name for the script you want to create, and then click **OK**. The new script name appears on the selected **PrePost Script** panel tab and the button changes to the **Stop and Save PrePost Script** button.
5. In the Rocky UI, perform the steps of the task that you want to repeat.
6. When the steps are complete, from the **PrePost Script** panel, click the **Stop and Save PrePost Script** button.

See Also:

- About Creating and Using PrePost Scripts (p. 296)
- Manually Apply a Saved Script (p. 302)
- Automatically Apply Scripts Directly Before or After Processing (p. 303)
- Edit a Saved Script Using the Python Programming Language (p. 303)
- Copy a Script (p. 304)
- Remove a Script (p. 305)
- Rename a Script (p. 305)

2.7.3. Add an External Script File to the Scripts Directory

Use this procedure to add a script to the Scripts directory.

1. Copy the PY script (p. 1394) file that you want to add. Tip: In order for Rocky to recognize it and have it show within the **PrePost Script** panel, the script file name must begin with *script*. **For example:** *script_video settings.py* Note: A Rocky script not named in this exact way may still be used but will not be listed within the **PrePost Script** panel. Instead, these scripts must be launched from outside of Rocky. (See also Use the Command Line to Play a Script (p. 1313).)
2. Ensure the **PrePost Script** panel is displayed. (From the **Tools** menu, click **PrePost Script**.)

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

3. From the **PrePost Script** panel, do one of the following: - To add the script in a way that it can be shared across all other Rocky projects, ensure the **Scripts shared across projects** tab is selected. - To add the script in a way that can be used only for this project, ensure the **Project scripts** tab is selected.

4. Click the **Open PrePost Scripts Directory** button.

5. Into the file directory location that opens, paste the PY file you copied in step 1.

See Also:

- About Creating and Using PrePost Scripts (p. 296)
- Manually Apply a Saved Script (p. 302)
- Automatically Apply Scripts Directly Before or After Processing (p. 303)
- Edit a Saved Script Using the Python Programming Language (p. 303)
- Copy a Script (p. 304)
- Remove a Script (p. 305)
- Rename a Script (p. 305)

2.7.4. Manually Apply a Saved Script

1. Ensure that the script you want to apply has been saved to the correct directory. (See also Add an External Script File to the Scripts Directory (p. 301).)

2. Within the Rocky UI, select the component to which you want the recorded steps of the script applied.

3. From the **PrePost Script** panel, select the name of the script you want applied, and then click the **Playback PrePost Script** button. The recorded or scripted steps are taken on the selected component. **Tip**: To see an overview of the script's processes, click the expand arrow to the left of the script's name. Click the arrow again to contract it.

Note: You can also have scripts applied automatically either directly before and/or after the simulation is processed. (See also Automatically Apply Scripts Directly Before or After Processing (p. 303).)

See Also:

- About Creating and Using PrePost Scripts (p. 296)
- Record a New Script (p. 301)
- Edit a Saved Script Using the Python Programming Language (p. 303)
- Copy a Script (p. 304)
- Remove a Script (p. 305)
- Rename a Script (p. 305)

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

2.7.5. Automatically Apply Scripts Directly Before or After Processing

1. Ensure that the scripts you want to automatically apply either directly before and/or after your simulation is processed are listed on the **Project scripts** tab of the **PrePost Script** panel. (See also Copy a Script (p. 304).)
2. From the **Data** panel, expand the **Solver** entity, and then click **PrePost Scripts**.
3. From the **Data Editors** panel, do the following:
   a. In the **Run before simulation** box, enable all the scripts you want to automatically run directly before the simulation is processed.
   b. In the **Run after simulation** box, enable all the scripts you want to automatically run directly after the simulation is processed.

Tip: You can also manually apply scripts at any time during your Rocky project. (See also Manually Apply a Saved Script (p. 302).)

See Also:

- About Creating and Using PrePost Scripts (p. 296)
- About Starting a Simulation (p. 908)

2.7.6. Edit a Saved Script Using the Python Programming Language

1. From the **PrePost Script** panel, select the tab containing the script you want to edit.
2. Click the **Open PrePost Scripts Directory** button.
3. From the directory dialog, right-click the script file you want to edit, open it with your favorite Python editor, modify it as desired, and then save the file as a PY extension. **Notes**:
   - In order for Rocky to recognize it and have it show within the PrePost Script panel, the script file name must begin with `script_`. **For example:** `script_video settings.py` A Rocky script not named in this exact way may still be used but will not be listed within the PrePost Script panel. Instead, these scripts must be launched from outside of Rocky. (See also Use the Command Line to Play a Script (p. 1313).)
   - To work properly in this version of Rocky, the Python 3 programming language must be used. (See also My old script no longer works (p. 1375).)

Tip:

Scripts also have access to the Rocky's PrePost Scripting functionality, which can be explored in the following ways:

- By using the Python Shell panel. (From the **Tools** menu, click **Python Shell**.) (See also Customize and Extend Rocky with Python (p. 294).)

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

- By using the **Contents**, **Search**, or **Index** tabs within the PrePost Scripting Manual. (From the Rocky **Help** menu, point to **Manuals** and then click **PrePost Scripting Manual**.)

See Also:

- About Creating and Using PrePost Scripts (p. 296)
- Record a New Script (p. 301)
- Add an External Script File to the Scripts Directory (p. 301)
- Manually Apply a Saved Script (p. 302)
- Automatically Apply Scripts Directly Before or After Processing (p. 303)
- Copy a Script (p. 304)
- Remove a Script (p. 305)
- Rename a Script (p. 305)

2.7.7. Copy a Script

1. Ensure the **PrePost Script** panel is displayed. (From the **Tools** menu, click **PrePost Script**.)
2. Ensure also that you have saved your Rocky project. (See also Save a Simulation Project (p. 852).)
3. From the **PrePost Script** panel, do one of the following:
   - To copy a shared script to the current project folder, select the **Scripts shared across projects** tab. Tip: Follow this procedure if you want to do either or both of the following:
     - Automatically apply a script either directly before and/or after the project is processed. (See also Automatically Apply Scripts Directly Before or After Processing (p. 303).)
     - Include the script in your Rocky Archive. (See also Archive a Simulation Project (p. 1294).)
   - To copy a current project script to the folder shared with all other projects, select the **Project scripts** tab.
4. Right-click the name of the script you want to copy, and then click **Copy Prepost Script to project**. The script you selected now appears on the other PrePost Script panel tab. In addition, any copies on the **Project scripts** tab now also appear on the **Solver PrePost Scripts** list. (See also Automatically Apply Scripts Directly Before or After Processing (p. 303).)

See Also:

- About Creating and Using PrePost Scripts (p. 296)
- Record a New Script (p. 301)
- Add an External Script File to the Scripts Directory (p. 301)

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

- Manually Apply a Saved Script (p. 302)
- Edit a Saved Script Using the Python Programming Language (p. 303)
- Remove a Script (p. 305)
- Rename a Script (p. 305)

2.7.8. Remove a Script

1. From the **PrePost Script** panel, select the tab containing the script you want to remove.
2. Right-click the name of the script you want to remove, and then click **Remove**. The script name is removed from the **PrePost Script** panel. The script PY file is also removed from the Script directory. If the script was removed from the **Project scripts** tab, then it was also removed from the **Solver** | **PrePost Scripts** boxes. (See also Automatically Apply Scripts Directly Before or After Processing (p. 303).)

See Also:

- About Creating and Using PrePost Scripts (p. 296)
- Record a New Script (p. 301)
- Add an External Script File to the Scripts Directory (p. 301)
- Manually Apply a Saved Script (p. 302)
- Automatically Apply Scripts Directly Before or After Processing (p. 303)
- Edit a Saved Script Using the Python Programming Language (p. 303)
- Copy a Script (p. 304)
- Rename a Script (p. 305)

2.7.9. Rename a Script

Note:

Renaming a script affects only what is shown in the Rocky UI. It does not change the PY file name that is saved in the PrePost Scripts directory.

---

1. From the **PrePost Script** panel, select the tab containing the script you want to rename.
2. Right-click the name of the script you want to rename, and then click **Rename**.
3. From the **Renaming PrePost Script** dialog, enter a new name for the script, and then click **OK**. The script's new name appears in the **PrePost Script** panel. If the renamed script was on the **Project scripts** tab, the name in the **Solver** | **PrePost Scripts** boxes are also updated accordingly. (See also Automatically Apply Scripts Directly Before or After Processing (p. 303).)

See Also:

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

- About Creating and Using PrePost Scripts (p. 296)
- Record a New Script (p. 301)
- Add an External Script File to the Scripts Directory (p. 301)
- Manually Apply a Saved Script (p. 302)
- Automatically Apply Scripts Directly Before or After Processing (p. 303)
- Edit a Saved Script Using the Python Programming Language (p. 303)
- Copy a Script (p. 304)
- Remove a Script (p. 305)

2.8. Data Collection

In Rocky, some data is collected automatically and some data you must first opt in to collecting before you process your simulation. In addition, some data is always collected right from the start of the simulation, and some data waits to be collected until passing certain thresholds that you define.

Use the topics below to help you understand what, how, and when data is collected in Rocky.

WHAT WOULD YOU LIKE TO DO?

- Collecting Data in Rocky (p. 306)
- Monitoring Overlaps (p. 309)
- Collecting Data on Boundaries (p. 312)
- Collecting Data on Particles (p. 319)
- Collecting Data on Contacts and Energy Spectra (p. 350)

See Also:

- Getting Started (p. 73)

COLLECTING DATA IN ROCKY

Rocky provides many options for calculating, collecting, and analyzing different kinds of data.

Since calculating data you do not want can slow down your processing, and retaining data you do not need can take up valuable storage space, Rocky requires you to opt-in (or turn on) certain kinds of data collection. It does this to help you maximize your computing power and storage space.

To help you better understand how and when data is calculated, use the table below and the topics that follow.

Table 1: Rocky data and statistics collection matrix

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

Type of Data or Statistic	How Collected	When Collected	See Also
Affecting Geometries			
Surface wear modification	Opt-in via the **Use Wear** checkbox (located on the **Geometry** | **Wear** tab) prior to processing.	After Wear **Start** value is reached.	Enable and View Surface Wear Modification on an Imported Geometry
Collision effects on belts and boundaries as a result of particles interactions	Opt-in via the **Boundary Collisions Statistics** Module prior to processing.	Immediately	Enable and View Collision Statistics for Boundaries
Belt or boundary motions	Automatic collection	Immediately	About Curves
All other data related to geometries	Automatic collection	Immediately	About Properties, About Curves
Affecting Particles			
Particles energy spectra	Enable the **Particles Energy Spectra** module prior to processing.	After the module's **Start Time** value is reached.	Enable and View Data for Particles Energy Spectra
Particle breakage	Automatic collection	After Breakage **Start** value is reached.	Enable and View Particle Breakage
Fluid effects upon individual particles	Opt-in via the **CFD Coupling Particle Statistics** Module prior to processing.	After CFD Coupling **Start Time** value is reached	Enable and View Fluid-Related Statistics for Particles
Collision effects upon the surface of a Particle set	Opt-in via the **Intra-particle Collisions Statistics** Module prior to processing.	Immediately	Enable and View Collision Statistics for Particle Surfaces (Intra)
Collision effects upon each particle-particle and particle-boundary pair group within the simulation	Opt-in via the **Inter-group Collision Statistics** Module prior to processing.	Immediately	Enable and View Inter-group Collision Statistics for Particles
Collision effects upon individual particles resulting from interactions with other particles and boundaries	Opt-in via the **Inter-particle Collision Statistics** Module prior to processing.	Immediately	Enable and View Collision Statistics for Particles Between Other Particles (Inter)

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

Type of Data or Statistic	How Collected	When Collected	See Also
Energy data for all Particle sets in the simulation	Opt-in via the **Particle Instantaneous Energies** Module prior to processing.	Immediately	Enable and View Instantaneous Energies for Particles
All other data related to particles	Automatic collection	Immediately	About Properties, About Curves
Affecting Contacts			
Contacts energy spectra	Enable the **Contacts Energy Spectra** module prior to processing.	After the module's **Start Time** value is reached.	Enable and View Data for Contacts Energy Spectra
Stress-related data (for analyzing via Eulerian Statistics User Process)	Opt-in via the **Collect Contacts Data** checkbox on the **Contacts** entity prior to processing.	Immediately	Turn On Contacts Data Collection
All other data related to Contacts	Opt-in via the **Collect Contacts Data** checkbox on the **Contacts** entity prior to processing.	Immediately	Turn On Contacts Data Collection, About Contacts
Affecting all other Rocky entities			
All other data related to all other entities	Automatic collection	Immediately	About Properties, About Curves

Note:

If you have defined a Multiple Element (meshed (p. 1392)) particle, what you have set for **Meshed Particles Upscaling** affects whether and how some post-processing properties and curves are displayed. (See also About Meshed Particles Upscaling (p. 1033).)

No matter how or when the data you want is collected, the resulting Properties (see also About Properties) can be visualized after processing in a view window and/or analyzed through a plot or histogram window. In addition, any resulting Curves (see also About Curves (p. 989)) can be analyzed through a plot or histogram window.

If you cannot see any of your simulation data and you know you have both collected it and have completed as much of the processing for which you want to have data, then you most likely have not yet created a view, plot, or histogram window by which to view and/or analyze the data or statistics you have collected.

Tip: Once your plot or histogram window is created (see also Graph Data within Rocky by Creating a Plot or Histogram (p. 1124)), you can then modify the display to your liking (see also About Changing the

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

Appearance of a Graph (Plot or Histogram) (p. 1126)), export the data into a CSV file for further analysis outside of Rocky (see also Export Data into a CSV File (p. 1164)), and more.

See Also:

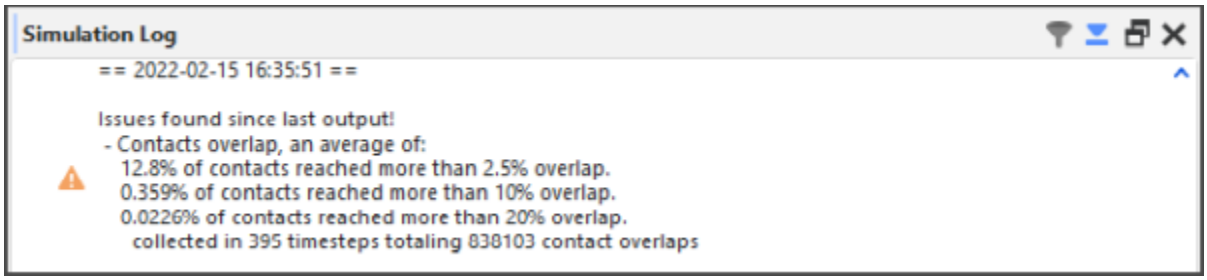
- Understand Data Collection in Rocky (p. 306)
- About 3D View Windows (p. 1040)
- Graphing (Plot or Histogram) a Data Set Within Rocky (p. 1101)
- About Properties (p. 933)
- About Curves (p. 989)
- About Meshed Particles Upscaling (p. 1033).

MONITORING OVERLAPS

In Rocky versions prior to 2022 R1, there was a built-in contact overlap monitor whose purpose was to check each contact (p. 1390) pair (particle-particle or particle-boundary) for the amount that they overlapped, the percentage of which was determined by the size of the smallest particle in the contact pair.

During simulation processing, if at any a contact pair's overlap exceeded one of three warning levels fixed at 2.5%, 10%, and 20%, a **Contacts overlap** message would be raised on the Simulation Log panel (Figure 1). (See also I get warnings or errors on my Simulation Log Panel (p. 1346).)

Figure 2.184: Example Contacts overlap message on the Simulation Log panel



Monitoring your contacts for the size of their overlaps is important because Rocky is based on the soft-sphere approach, which uses the overlap value in order to compute collision forces. For example, in a typical simulation overlaps of 2.5% might be considered acceptable; however, simulations with overlaps larger than 10-20% should probably not be trusted as overlap values that large can lead to serious stability and accuracy issues.

Although it is a good practice to monitor overlap levels to guarantee they are below the desired values for a DEM solver, for some types of projects, the default warning levels Rocky uses might not be appropriate; or, it might be desired for there to be no overlap checks at all.

Therefore, as of Rocky 2022 R1, you are able to choose:

- Whether or not you want contact overlaps to be monitored. Note: The monitor is "on" by default to match the behavior in older versions of Rocky.

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

- What three overlap levels about which you want to be warned. Note: The initial values are set as to match the warning levels used in older versions of Rocky.

These tasks are accomplished via an embedded Module (p. 1390) called **Contacts Overlap Monitor** (Figure 2).

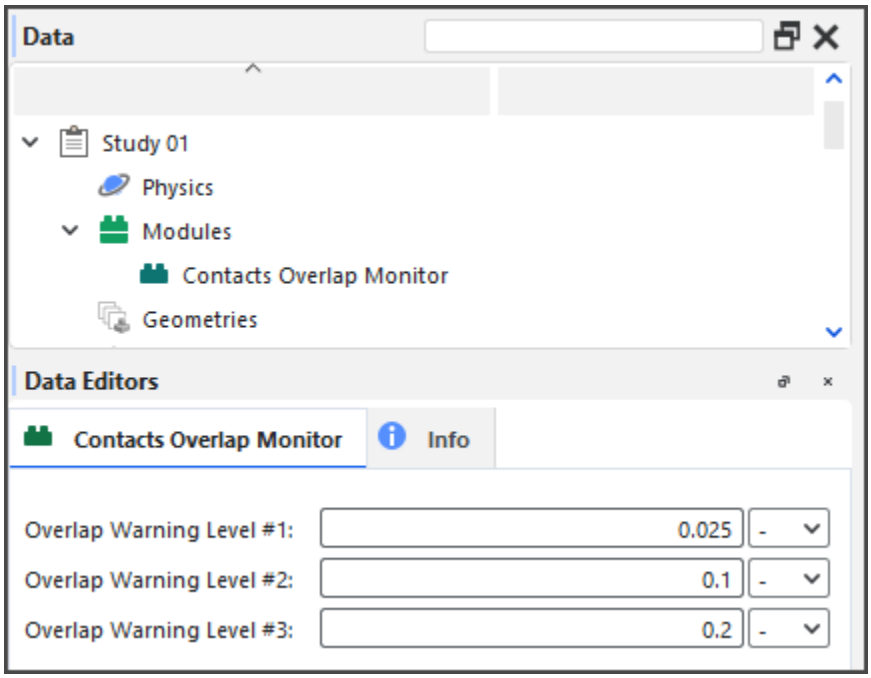


Figure 2: Options in the Data Editors panel for the Contacts Overlap Monitor Module

CONTACTS OVERLAP MONITOR OPTIONS

Use Figure 2 above and the table below to understand how to monitor your contact overlaps.

Table 1: Modules, Contacts Overlap Monitor parameter definitions

Setting	Description	Range
Overlap Warning Level #1	Defines the value of the first (of three) overlap warning thresholds. If a contact pair's overlap-the percentage of which is based upon the size of the smallest particle in the contact pair-exceeds this value, a message will be raised on the Simulation Log panel. (See also About the Simulation Log Panel.) **Note:** The default value of 2.5% matches the built-in overlap used in prior versions of Rocky.	Any value
Overlap Warning Level #2	Defines the value of the second (of three) overlap thresholds. If a contact pair's overlap-the percentage of which is based upon the size of the smallest particle in the contact pair-exceeds this value, a	Any value

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

Setting	Description	Range
	warning will be raised on the Simulation Log panel. (See also About the Simulation Log Panel.) **Note:** The default value of 10% matches the built-in overlap used in prior versions of Rocky.	
Overlap Warning Level #3	Defines the value of the third (of three) overlap thresholds. If a contact pair's overlap-the percentage of which is based upon the size of the smallest particle in the contact pair-exceeds this value, a warning will be raised on the Simulation Log panel. (See also About the Simulation Log Panel.) **Note:** The default value of 20% matches the built-in overlap used in prior versions of Rocky.	Any value

What would you like to do next?

- Learn more About the Contacts Overlap Monitor Module (p. 872)
- Turn off the Overlap Monitor (p. 311)
- Change the Overlap Monitor Warning Levels (p. 312)

TURN OFF THE OVERLAP MONITOR

The **Contacts Overlap Monitor** module is "on" (enabled) by default to match the behavior in older versions of Rocky.

Although it is a good practice to monitor overlap levels to guarantee they are below the desired values for a DEM solver, use the following procedure if you do not want to monitor contact overlaps in your simulation.

Note:

This process can only be completed prior to processing your simulation.

To Turn Off the Overlap Monitor:

1. From the **Data** panel, select **Modules**.
2. From the **Data Editors** panel, clear the **Contacts Overlap Monitor** checkbox.

See Also:

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

- About Monitoring Overlaps (p. 309)
- Change the Overlap Monitor Warning Levels (p. 312)
- About Rocky Modules (p. 221)

CHANGE THE OVERLAP MONITOR WARNING LEVELS

The initial values defined in the **Contacts Overlap Monitor** module are designed to match the warning levels used in older versions of Rocky. Use the following procedure if you want to define different warning thresholds for your contact overlaps.

Note: This process can only be completed prior to processing your simulation.

To Change the Warning Level Values used in the Overlap Monitor:

1. From the **Data** panel, under **Modules**, select the **Contacts Overlap Monitor** entity.
2. From the **Data Editors** panel, enter the values you want for the three separate **Overlap Warning Level** fields.

See Also:

- About Monitoring Overlaps (p. 309)
- Turn off the Overlap Monitor (p. 311)
- About Rocky Modules (p. 221)

COLLECTING DATA ON BOUNDARIES

In Rocky, some boundaries data is collected automatically and some data you must first opt in to collecting before you process your simulation. In addition, some boundaries data is always collected right from the start of the simulation, and some data waits to be collected until passing certain thresholds that you define.

Use the topics below to help you understand what, how, and when boundaries data is collected in Rocky.

COLLISION STATISTICS FOR BOUNDARIES

By default, Rocky automatically collects data about the surface, make-up, and movement of individual geometries and conveyors. However, if you want to analyze the effects of particles colliding with these boundaries (for example, to study collision frequency, intensities, impact velocities, and so on), you will need to turn on Boundary Collision Statistics collection prior to processing your simulation.

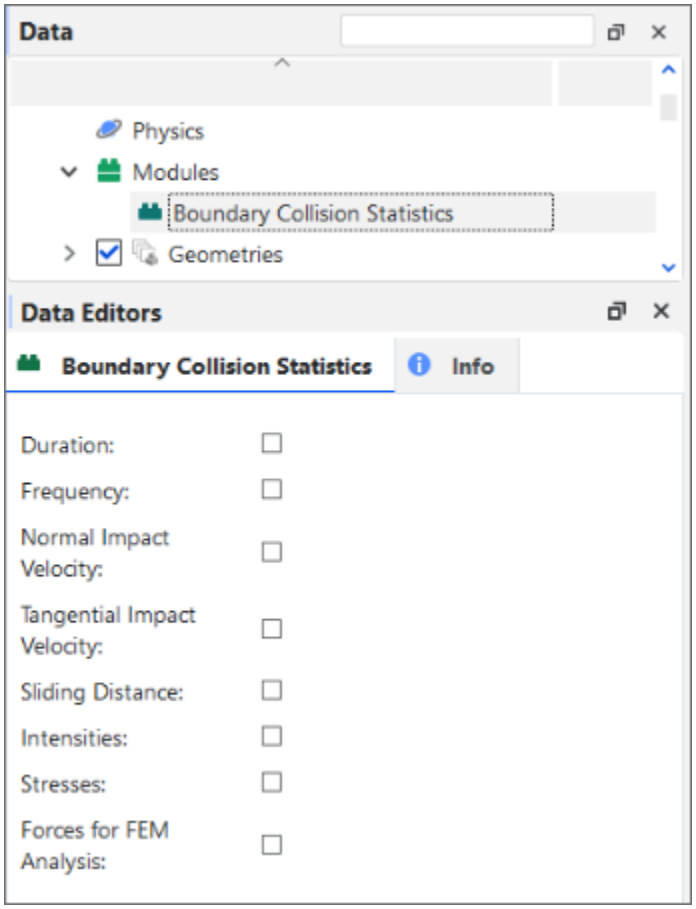
About Collecting Boundary Collision Statistics

If you want to analyze this type of information in your simulation, you must first enable its collection via its module (p. 1392) prior to processing your simulation.

Because collecting collisions statistics can take more processing time, memory, and disk storage due to increased file sizes, Rocky enables you to select from one or more sub-categories of statistics to collect. These are made available through the **Boundary Collisions Statistic** Module (Figure 1).

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

Figure 2.185: Options in the Data Editors panel when the Boundary Collision Statistics Module is enabled



Tip: To maximize your processing capabilities, choose only the statistics that you require for your analyses.

During processing, Rocky collects the selected collision statistics between two consecutive output time levels for all boundaries in your project.

Tip: To view walk-through examples of collecting and analyzing boundary collisions statistics, refer to the following Tutorials:

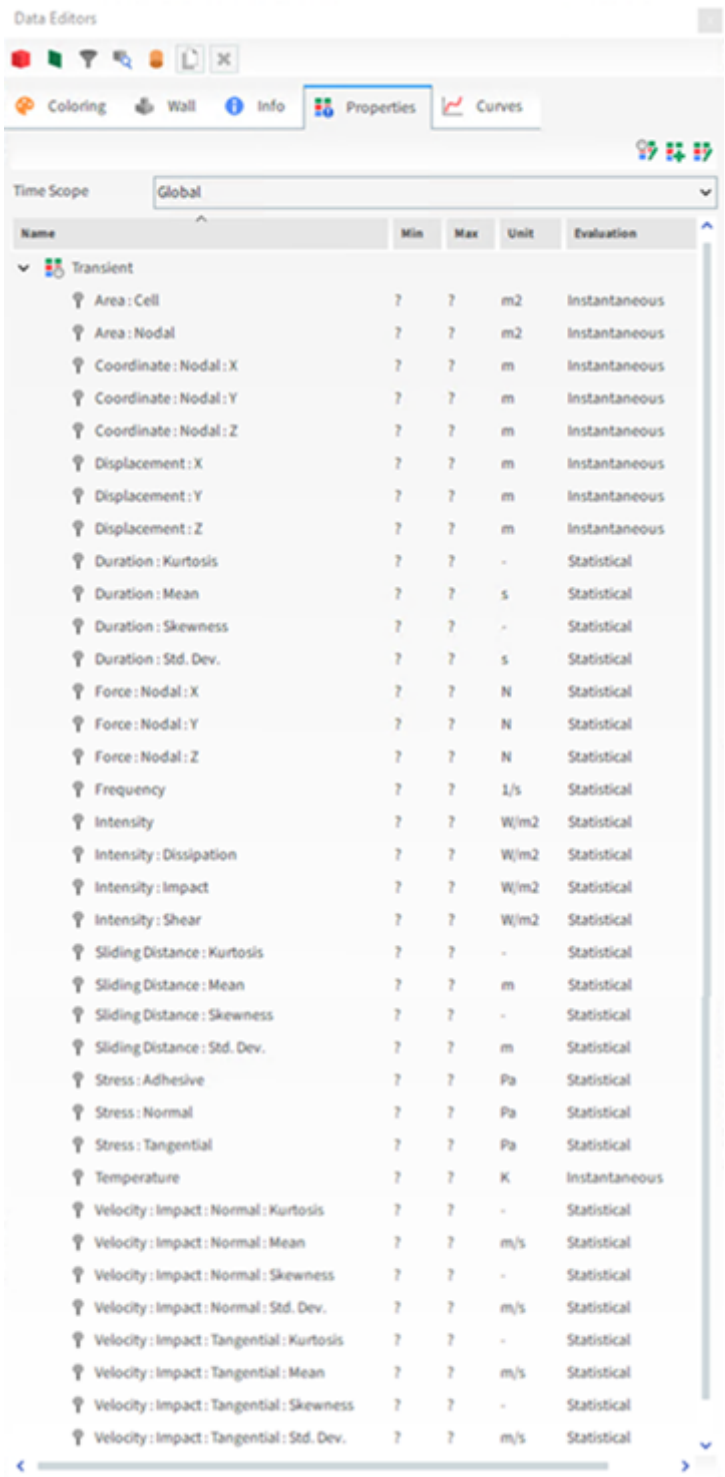
- Tutorial - High Pressure Grinding Roll (HPGR) in the *Rocky Tutorial Guide*
- Tutorial - Static Structural in the *Rocky Tutorial Guide*
- Tutorial - Transient Structural in the *Rocky Tutorial Guide*
- Tutorial - Mixing Tee in the *Rocky Tutorial Guide*
- Tutorial - Backhoe Loader in the *Rocky Tutorial Guide*

About Analyzing Boundary Collision Statistics

Once the collisions data is collected (post-processing), specific collisions-related Properties (Figure 2) and Curves (Figure 3) will be available for the geometry or conveyor you select. (See also About Properties (p. 933) and About Curves (p. 989).) The specific Properties and Curves available depends upon which statistics you enabled prior to processing.

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

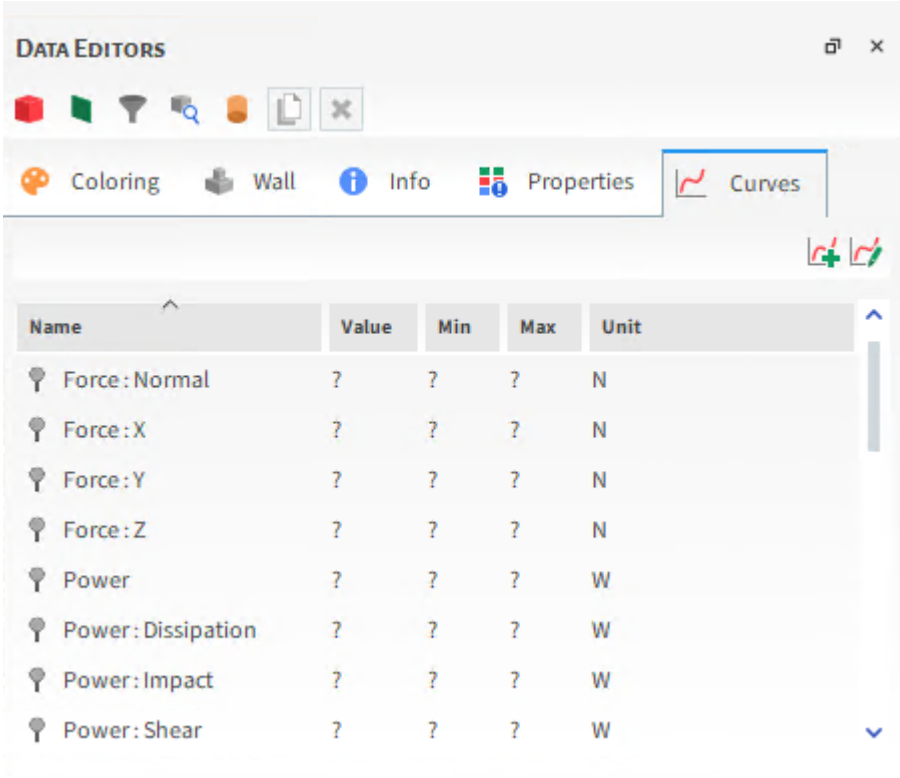
Figure 2.186: Boundary Collision Properties available for geometries after processing



Tip: These properties will be categorized as **Statistical** in the **Evaluation** column. (See also About Viewing an Individual Statistic (p. 1098).)

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

Figure 2.187: Boundary Collision Curves available for geometries after processing

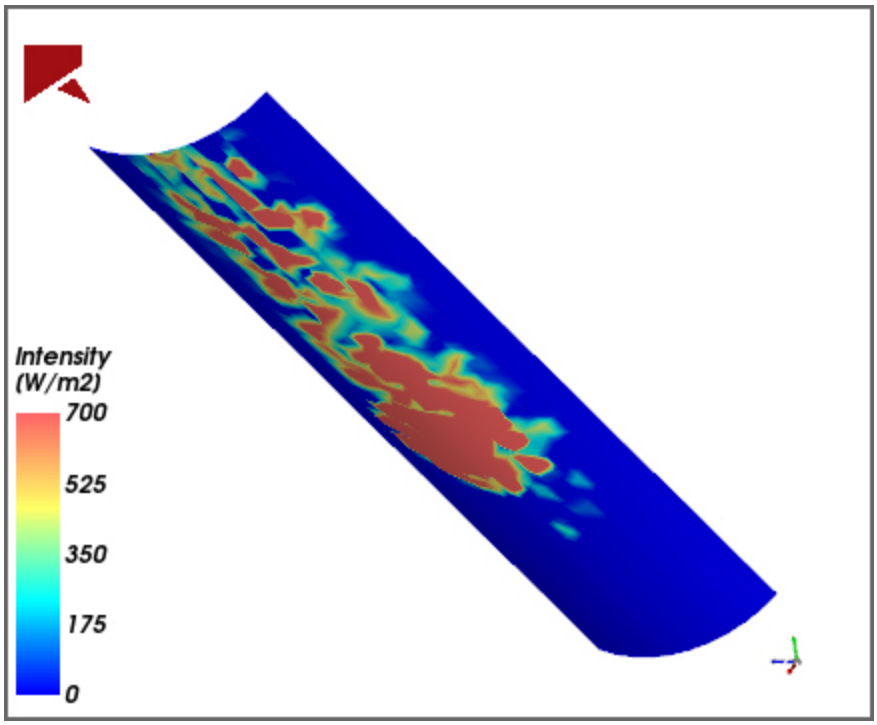


You can then choose to analyze the resulting Properties or Curves in a plot or histogram window. (See also Graphing (Plot or Histogram) a Data Set Within Rocky (p. 1101))

Or, you can then choose to display the Properties information graphically on the surface of your boundaries in a 3D View window. This can be useful, for example, for showing a color map of surface intensity (Figure 3). (See also View a Color Map of Wear on the Default Belt or Imported Geometry Itself (p. 381).)

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

Figure 2.188: Color map of surface intensity



By using the **Properties** tab for the geometry, the options on the **Coloring** tab, and/or by using the slider on the Time toolbar (see also About the Time Toolbar (p. 1279)), you can change how the data appears in the window. **Tip**: You may also limit your data further by using Time Statistics Properties. (See also About Adding and Editing Time Statistics Properties (p. 1023).)

Tip: Learn more by referring to the "Collision statistics" section in the DEM Technical Manual. (From the Rocky **Help** menu, point to **Manuals**, and then click **DEM Technical Manual**.)

Boundary Collision Statistics Collection Options

Use Figure 1 above and the table below to understand how to collect collision statistics for your boundaries.

Table 1: Modules, Boundary Collision Statistics parameter definitions

Setting	Description	Range
Duration	When enabled, Rocky will collect the mean, standard deviation, skewness, and kurtosis values of the duration of the collisions recorded in different regions of the geometry, during an interval between two consecutive output times. This can be useful when you are able to relate the duration to the amount of mass or heat transferred, for instance, in simulations involving chemical reactions and/or heat transfer.	Turns on or off
Forces for FEM Analysis	When enabled, Rocky will collect the average force in various directions for each individual geometry node. This can be useful for analyzing directional forces, or side thrust.	Turns on or off

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

Setting	Description	Range
Frequency	When enabled, Rocky will collect the average collision frequency recorded in different regions of the geometry, during an interval between two consecutive output times. This can be useful for analyzing shot flow, or for understanding the distribution of the frequency of the collisions against the surface triangles.	Turns on or off
Intensities	When enabled, Rocky will collect the average dissipation and impact power values measured by each individual geometry triangle. This can be useful for analyzing impact wear or power draw.	Turns on or off
Normal Impact Velocity	When enabled, Rocky will collect the mean, standard deviation, skewness, and kurtosis values of the impact relative velocity in the normal direction resulting from the collisions recorded in different regions of the geometry, during an interval between two consecutive output times. This can be useful for analyzing impact wear.	Turns on or off
Sliding Distance	When enabled, Rocky will collect the mean, standard deviation, skewness, and kurtosis values of the sliding distance, which is the distance that a particle moves during a collision, parallel to the boundary triangle plane where the collision occurs. This can be useful for analyzing shear wear.	Turns on or off
Stresses	When enabled, Rocky will collect the adhesion (if applicable; see note), normal and tangential stress values measured by each individual geometry triangle. This can be useful for analyzing the distribution of load due to particle collisions on a geometry. **Note:** Adhesive stresses are collected only when an **Adhesive Force** model (other than **None**) is enabled (see also About Physics Parameters), and an Adhesive **Force Fraction** for a boundary-boundary and/or boundary-particle Materials Interaction pair has a value set higher than zero (see also About Modifying Materials Interactions and Adhesion Values).	Turns on or off
Tangential Impact Velocity	When enabled, Rocky will collect the mean, standard deviation, skewness, and kurtosis values of the impact relative velocity in the tangential direction resulting from the collisions recorded in different regions of the geometry, during an interval between two consecutive output times. This can be useful for analyzing impact wear.	Turns on or off

What would you like to do next?

- Enable and View Collision Statistics for Boundaries (p. 318)
- Learn more About the Boundary Collision Statistics Module (p. 868)
- Learn more about how to Add and Edit Geometry Components (p. 480)

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

- Learn more About Collecting Data in Rocky (p. 306)

Enable and View Collision Statistics for Boundaries

Analyzing collision statistics for boundaries (see also About Collision Statistics for Boundaries (p. 312)) involves turning on the collection of the collision properties you want prior to processing your simulation.

After processing, you are then able to analyze the resulting Properties and/or Curves as you normally would.

To Enable and View Collision Statistics for Boundaries:

1. Set up the simulation as you normally would. (See also Set Simulation Parameters (p. 414).)
2. Before processing your simulation, do all of the following:
   a. From the **Data** panel, select **Modules**, and then from the **Data Editors** panel, select **Boundary Collision Statistics**.
   b. From the **Data** panel, under **Modules**, select the new **Boundary Collision Statistics** entry.
   c. From the **Data Editors** panel, select the checkboxes for the type of statistics you want collected. (See also About Collision Statistics for Boundaries (p. 312).)
   d. Verify that the default conveyor or imported geometry has its **Triangle Size** set small enough to enable the detail you want. (0.1 m is recommended for most chutes and mills). (From the **Data** panel under **Geometry**, select the component you want to verify. From the **Data Editors** panel, on the **Geometry** sub-tab, verify the **Triangle Size** value.)
3. Process the simulation as you normally would. (See also Processing a Simulation (p. 907).)
4. From the **Data** panel, under **Geometries**, select the component that you want to analyze.
5. From the **Data Editors** panel, select either the **Properties** or the **Curves** tab.
6. Create a plot (see also Graph Data within Rocky by Creating a Plot or Histogram (p. 1124)) or visualize the data in a 3D View window (see also About 3D View Windows (p. 1040)). Tips:
   - With Properties, you can also choose to View a Color Map of Wear on the Default Belt or Imported Geometry Itself (p. 381).
   - You may also limit your data further by using Time Statistics Properties. (See also About Adding and Editing Time Statistics Properties (p. 1023).)

See Also:

- About Collecting Data in Rocky (p. 306)
- About Collision Statistics for Boundaries (p. 312)
- About the Boundary Collision Statistics Module (p. 868)
- About Properties (p. 933)
- About Curves (p. 989)

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

COLLECTING DATA ON PARTICLES

In Rocky, some Particles data is collected automatically and some data you must first opt in to collecting before you process your simulation. In addition, some Particles data is always collected right from the start of the simulation, and some data waits to be collected until passing certain thresholds that you define.

Use the topics below to help you understand what, how, and when Particles data is collected in Rocky.

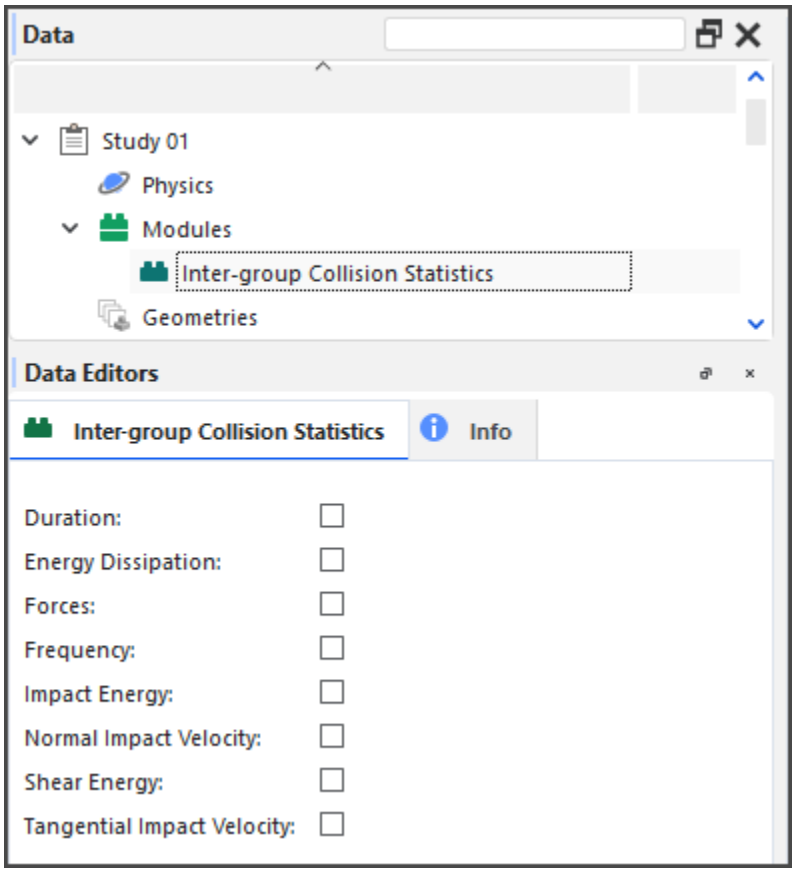
ABOUT INTER-GROUP CCOLLISION STATISTICS

If you want to analyze the effects of collisions upon various particle-particle or particle-boundary pair groups within your simulation, you can choose to collect Inter-group Collision Statistics prior to processing your simulation.

About Collecting Inter-group Collision Statistics

Because collecting collisions statistics can take more processing time, memory, and disk storage due to increased file sizes, Rocky enables you to select from one or more sub-categories of statistics to collect. These are made available through the **Inter-group Collision Statistics** Module (Figure 1).

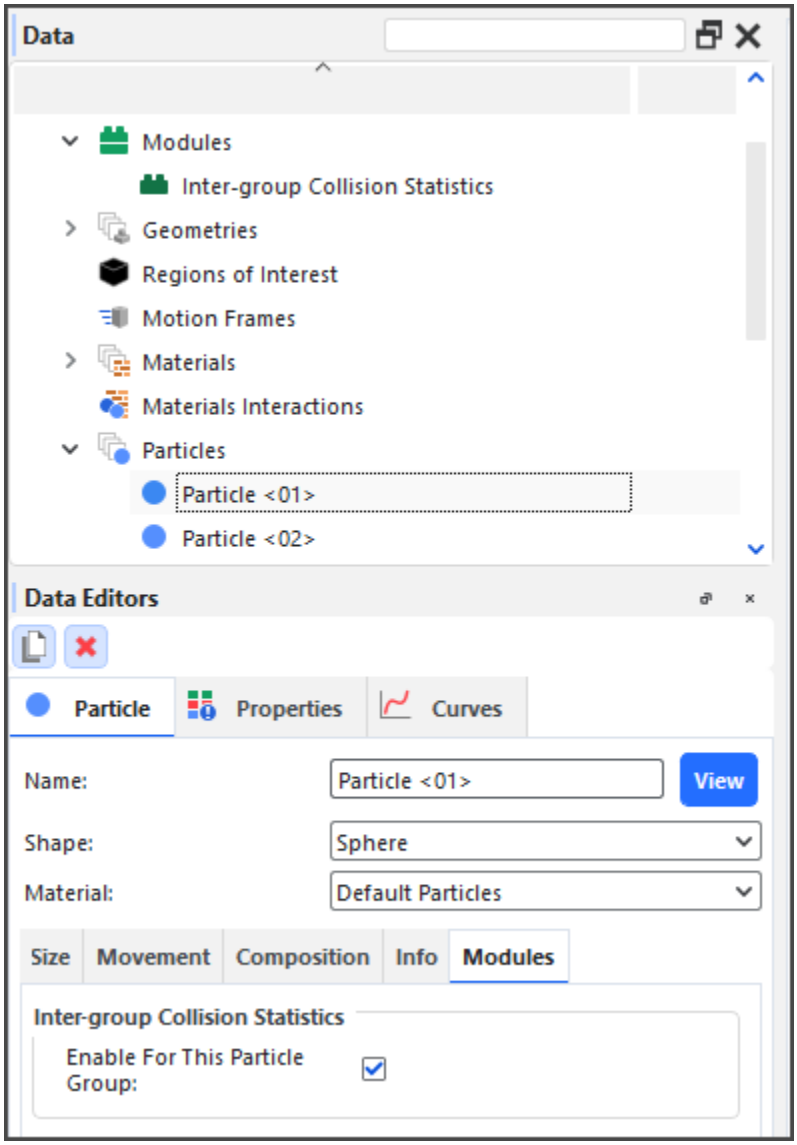
Figure 2.189: Options in the Data Editors panel when the Inter-group Collision Statistics Module is enabled



In addition, only particles and geometries belonging to particle groups and geometry components that have been enabled for Inter-group Collision Statistics collection will be recorded (Figures 2 and 3).

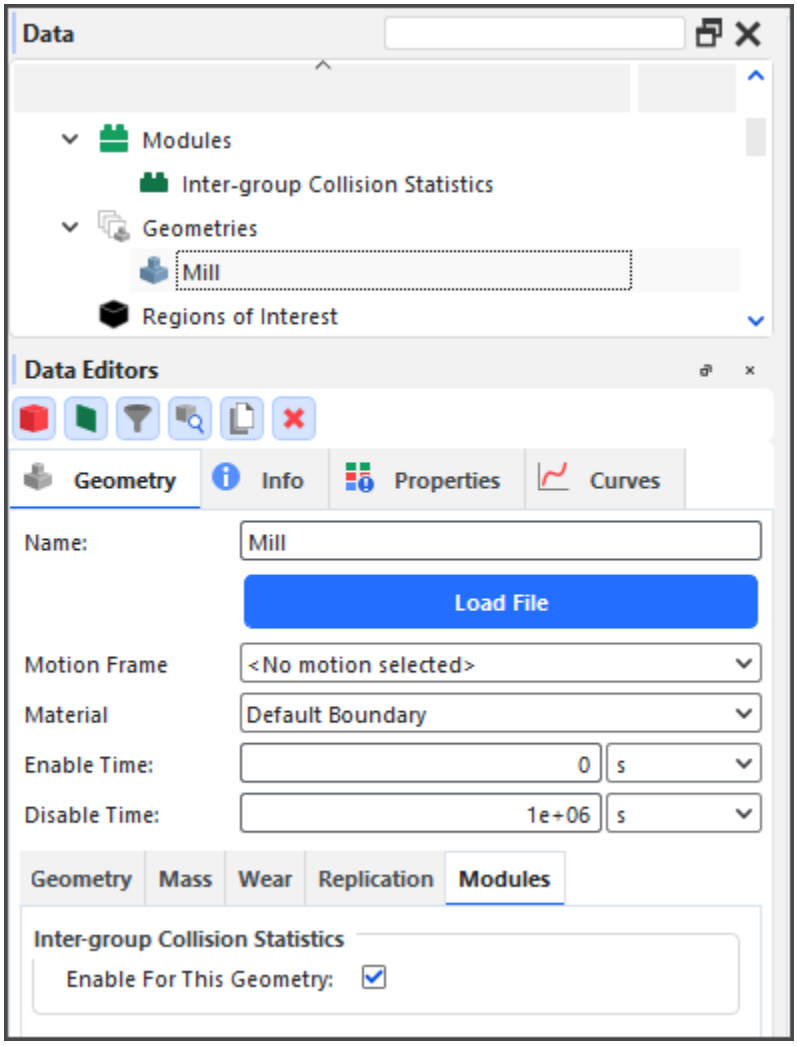
Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

Figure 2.190: Additional module options for Particle groups when the Inter-group Collision Statistics module is enabled



Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

Figure 2.191: Additional module options for a geometry component when the Inter-group Collision Statistics module is enabled



Tip:

To maximize your processing capabilities, choose only the statistics that you require for your analyses.

During processing, Rocky collects the selected collision statistics between two consecutive output time levels for each participating particle-particle and particle-boundary pair.

About Analyzing Inter-group Collision Statistics

Once the collisions data is collected (post-processing), specific collisions-related Curves (Figure 2) will be available for the main Particles entity. (See also About Curves (p. 989).)

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

Figure 2.192: Curves for Particles (simulation-wide) when Inter-group Collision Statistics are enabled

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

Note:

For all Inter-group Collisions Statistics curves, the effects of upscaling will be ignored for any meshed (p. 1392) particles with **Meshed Particles Upscaling** enabled. (See also About Meshed Particles Upscaling (p. 1033).)

You can then choose to analyze the resulting Curves in a time or cross plot window. (See also Graphing (Plot or Histogram) a Data Set Within Rocky (p. 1101).)

Tip:

Learn more by referring to the "Collision statistics" section in the DEM Technical Manual. (From the Rocky **Help** menu, point to **Manuals**, and then click **DEM Technical Manual**.)

Inter-group Collision Statistics Collection Options

Use Figure 1 above and the table below to understand how to collect Inter-group Collision Statistics.

Table 1: Modules, Inter-group Collision Statistics parameter definitions

Setting	Description	Range
Duration	When enabled, Rocky will collect the mean, standard deviation, skewness, and kurtosis values of the duration of the collisions recorded for each particle-particle and particle-boundary pair in the simulation.	Turns on or off
Energy Dissipation	When enabled, Rocky will collect the energy dissipation values of the collisions recorded for each particle-particle and particle-boundary pair in the simulation.	Turns on or off
Forces	When enabled, Rocky will collect the average force in both the normal and tangential directions of the collisions recorded for each particle-particle and particle-boundary pair in the simulation.	Turns on or off
Frequency	When enabled, Rocky will collect the average frequency of the collisions recorded for each particle-particle and particle-boundary pair in the simulation.	Turns on or off
Impact Energy	When enabled, Rocky will collect impact energy values of the collisions recorded for each particle-particle and particle-boundary pair in the simulation.	Turns on or off
Normal Impact Velocity	When enabled, Rocky will collect the mean, standard deviation, skewness, and kurtosis values of the impact relative velocity in the normal direction of the collisions recorded for each particle-particle and particle-boundary pair in the simulation.	Turns on or off
Shear Energy	When enabled, Rocky will collect shear energy values of the collisions recorded for each particle-particle and particle-boundary pair in the simulation.	Turns on or off

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

Setting	Description	Range
Tangential Impact Velocity	When enabled, Rocky will collect the mean, standard deviation, skewness, and kurtosis values of the impact relative velocity in the tangential direction of the collisions recorded for each particle-particle and particle-boundary pair in the simulation.	Turns on or off

What would you like to do next?

- Learn more About the Inter-group Collision Statistics Module (p. 875)
- Enable and View Inter-group Collision Statistics for Particles (p. 345)
- Learn more About Adding and Editing Particle Sets (p. 602)
- Learn more About Collecting Data in Rocky (p. 306)

See Also:

- Collecting Data on Contacts and Energy Spectra (p. 350)

ABOUT INTRA-PARTICLE COLLISION STATISTICS

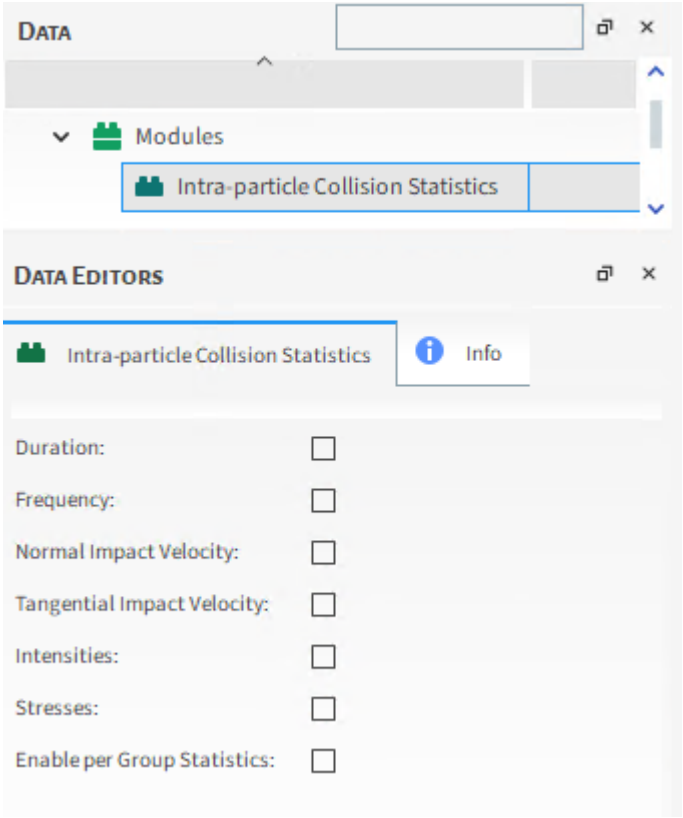
For certain solid and flexible Particle sets (p. 1392) in Rocky (see *Supported Particle Sets* section below), you can choose to have collision data between two consecutive output time levels collected by Rocky during the simulation. In this version of Rocky, this is referred to as *Intra-particle Collision Statistics*.

About Collecting Intra-particle Collision Statistics

Because collecting collisions statistics can take more processing time, memory, and disk storage due to increased file sizes, Rocky enables you to select from one or more sub-categories of statistics to collect. These are made available through the **Intra-particle Collision Statistics** Module (Figure 1).

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

Figure 2.193: Options in the Data Editors panel when the Intra-particle Collision Statistics Module is enabled



Tip:

To maximize your processing capabilities, choose only the statistics that you require for your analyses.

During processing, Rocky collects the selected collision statistics between two consecutive output time levels for all applicable particle sets in your project.

Tip: To see a walk-through example of collecting and analyzing Intra-particle collisions statistics, refer to Tutorial - Tablet Coater in the *Rocky Tutorial Guide*.

Supported Particle Sets

This feature works only with Particle sets that have the following characteristics:

- Single size (or monosized) only. (No particle size distributions (PSDs).)
- Unbroken shapes only. (No breakage modeling.)
- For Solids, Polyhedron or Custom Polyhedrons (either concave or convex) shapes only; can be either rigid or flexible, however. (No Spheres nor any type of Sphero-shapes; no Briquettes nor Faceted Cylinder shapes.)
- For Fibers, flexible shapes only. (No single-element Fiber compositions. No Shell shapes of any kind, neither rigid nor flexible.)

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

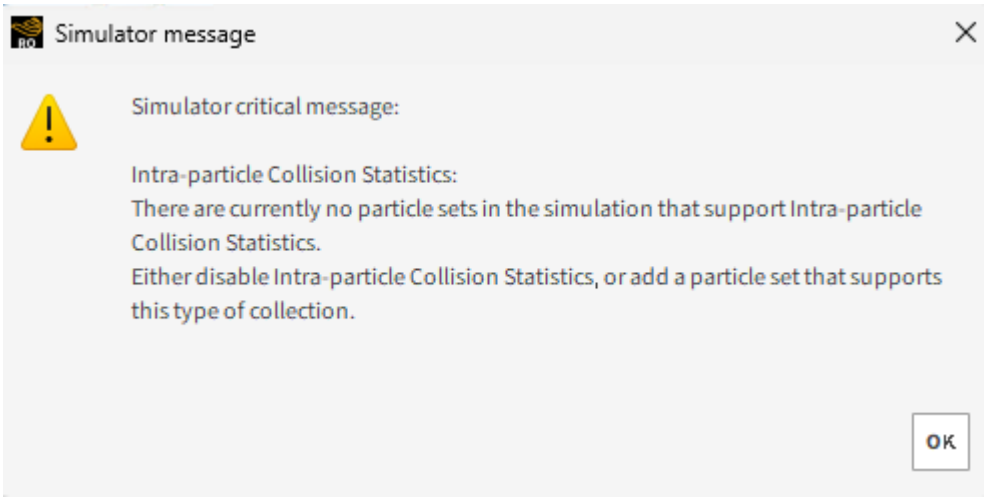
(See also About Adding and Editing Particle Sets (p. 602).)

Note:

When used with Multi-Element (meshed (p. 1392)) particles with **Meshed Particles Upscaling** enabled, Intra-particle Collision Statistics properties will be provided but the effects of upscaling will be ignored. This means that rather than providing data for the whole particle, data will be provided for each individual Element (p. 1390). (See also About Meshed Particles Upscaling (p. 1033).)

If you choose to enable Intra-particle Collision Statistics, then you must have at least one Particle set in your simulation that meets the above criteria. Otherwise, Rocky will not process the simulation, and you might see the following error:

Figure 2.194: Rocky error shown if no particle sets in the simulation support Intra-particle Collisions Statistics



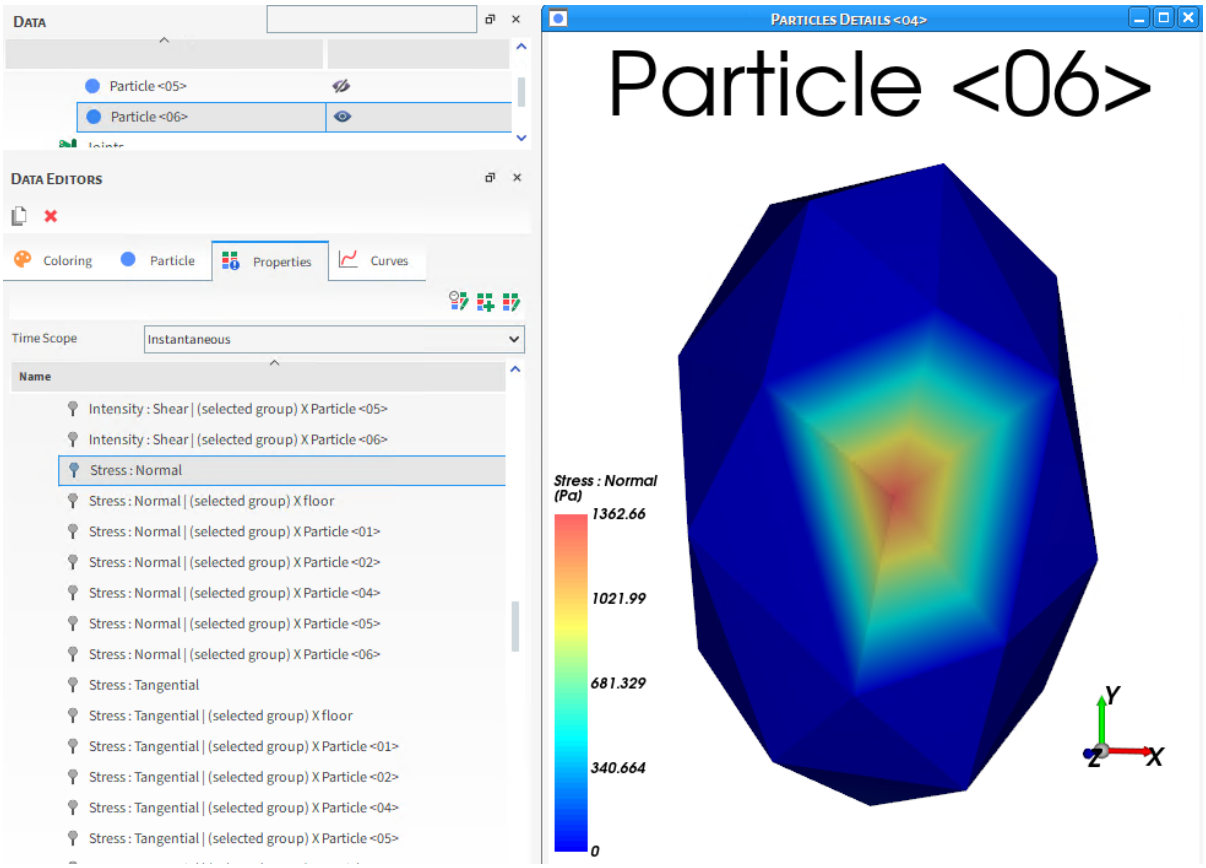
If you see this error after attempting to process your simulation, either disable Intra-particle Collision Statistics, or add a particle set that supports this type of collection.

About Analyzing Intra-particle Collisions Statistics

Once the collisions data is collected (post-processing), specific collisions-related Properties (Figure 3) will be available for the supported particle set you select. (See also About Properties (p. 933).) The specific Properties available depends upon which statistics you enabled prior to processing.

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

Figure 2.195: Particles Details window showing Stress Normal collision statistics for a particle set



Tip:

These properties will be categorized as **Statistical** in the **Evaluation** column. (See also About Viewing an Individual Statistic (p. 1098).)

Tip:

Although it is possible to visualize the data at any time during processing, it is recommended that you avoid analyzing the data until the simulation reaches the point during which the particles are nearly constant (steady state). This is because the resulting data will have little meaning if the number of particles has a lot of variation.

You can then choose to display the collisions data graphically on the surface of a representative particle in the **Particles Details** window (Figure 3).

By using the slider on the **Time** toolbar (see also About the Time Toolbar (p. 1279)), you can see how the data changes at different points in time.

Note:

Because the Thermal Model considers a uniform temperature per individual particle, collision statistics cannot account for temperature. As a result, the Temperature property will not be

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

available to view in a **Particles Details** window. However, in Collisions Statistics simulations that have both the Thermal Model enabled and use thermal-supported shape types (see also Particle and Input Limitations (p. 59)), you can still view the Temperature property from the main **Particles** entity, or from a **3D View** window. (See also Enable Thermal Modeling Calculations (p. 400).)

About Collision Statistics Visualization

In the collision statistics visualization, a value displayed at a given vertex is representative of all collisions in the region of influence of the vertex that occurred in all enabled particles of the particle set being analyzed, during the time lapse between two output time levels. This means that data displayed at time t_1 is the result of the statistics of the collision that happened between time $t_1 - \Delta t$ and t_1, where Δt is the output period.

As usual in these type of representations, values displayed at other, non-vertex points of the particle are obtained by interpolation of available values from the surrounding vertices.

In order to get a statistical value per particle, the collision statistics that happened during that output is divided by the number of enabled particles at the time when the statistics are displayed. Therefore, it is important that these statistics should be used only when you have reached a point during the simulation where the number of particles during that output is nearly constant.

Otherwise, for example, suppose you have 100 particles from the beginning of the output period to the middle of it, and at the last time step of the output period, you have only 1 particle. The statistics of all 100 particles that were there will be divided by 1 (the number of particles at the final time step of that output period), giving values much higher than expected.

Conversely, if you get these statistics at the beginning of your simulation and particles are entering, for example, you may have 0 particles at the beginning of the first output period and 1000 particles at the end. In this case, the collision statistics will be divided by 100 and will give values much smaller than expected.

Tip:

Learn more by referring to the "Collision statistics" section in the DEM Technical Manual. (From the Rocky **Help** menu, point to **Manuals**, and then click **DEM Technical Manual**.)

Tip:

To see a walk-through example of gathering and displaying collision statistics for particles, review Tutorial - Tablet Coater in the *Rocky Tutorial Guide*.

Intra-particle Collision Statistics Collection Options

Use Figure 1 above, and the figure and table below to understand how to collect Intra-particle Collisions Statistics.

Table 1: Modules, Intra-particle Collisions Statistics parameter definitions

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

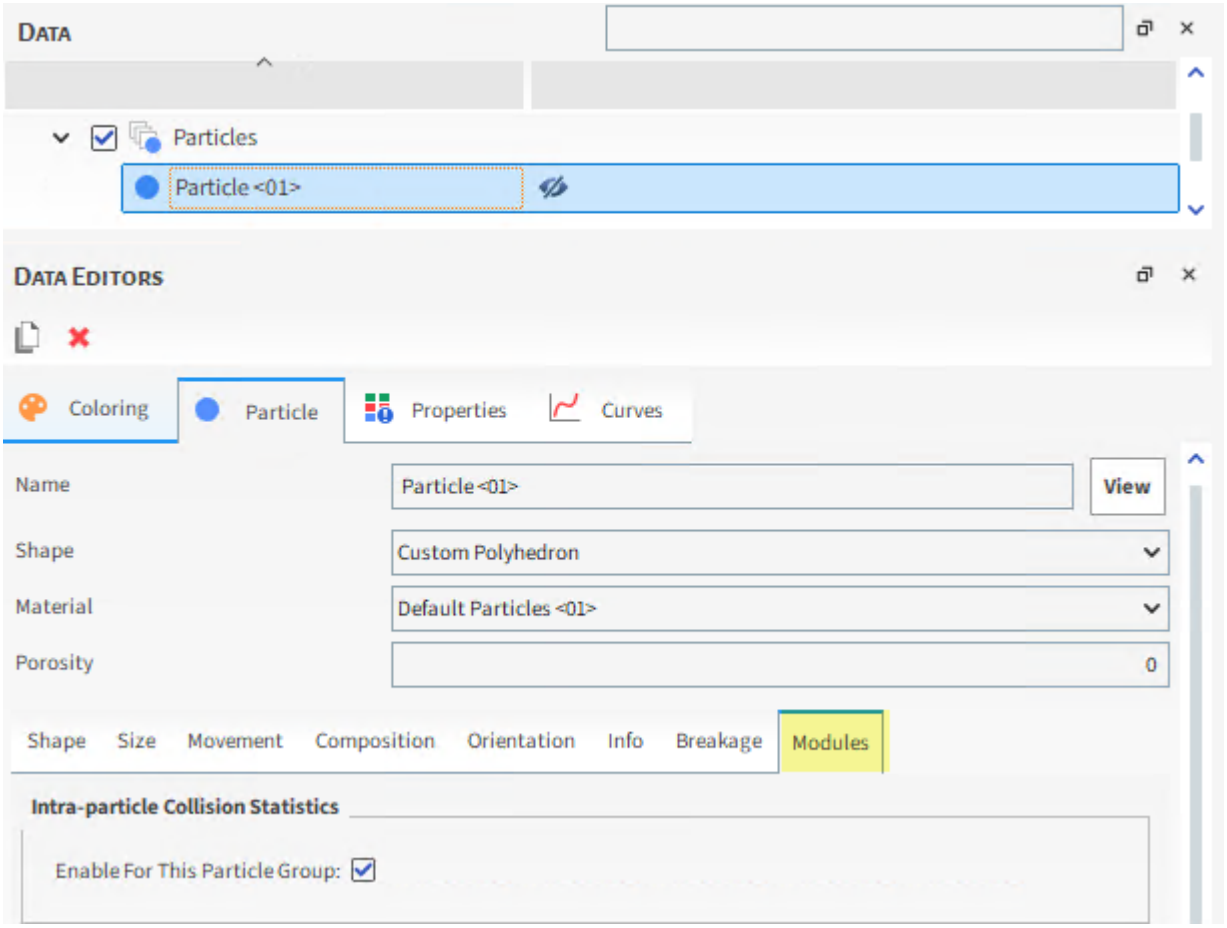
Setting	Description	Range
Duration	When enabled, Rocky will collect the mean, standard deviation, skewness, and kurtosis values of the duration of the collisions recorded in different regions of the representative particle of the selected particle set, during an interval between two consecutive output times. This can be useful when you are able to relate the duration to a certain process, such as in simulations involving chemical reactions and/or heat transfer.	Turns on or off
Frequency	When enabled, Rocky will collect the average collision frequency recorded in different regions of the representative particle of the selected particle set, during an interval between two consecutive output times. This can be useful for analyzing the distribution of the frequency of the collisions around the particle surface.	Turns on or off
Intensities	When enabled, Rocky will collect the power transferred per unit area in different regions of the representative particle of the selected particle set. Specifically: • The intensity value is related to the work made by the contact forces during a collision. • The impact work considers the work made by the normal force during the loading portion of the collision. • The dissipation work considers the entire collision cycle, including the unloading period (for either the normal or tangential forces). This can be useful for evaluating the impact work around the particle, perhaps to help avoid exaggerated wear in a certain region of the particle, for example.	Turns on or off
Normal Impact Velocity	When enabled, Rocky will collect the mean, standard deviation, skewness, and kurtosis values of the impact relative velocity in the normal direction resulting from the collisions recorded in different regions of the representative particle of the selected particle set, during an interval between two consecutive output times. This can be useful when analyzing particle breakage or granules de-agglomeration, for example.	Turns on or off
Stresses	When enabled, Rocky will collect the adhesion (if applicable; see note), normal and tangential stress values measured by different regions of the representative particle of the selected particle set. This can be useful for analyzing damage to the particle surface due the action of concentrated forces in certain regions. **Note:** Adhesive stresses are collected only when an **Adhesive Force** model (other than **None**) is enabled (see also About Physics Parameters), and an Adhesive **Force Fraction** for a particle-boundary and/or particle-particle Materials Interaction pair has a value set higher than zero (see also About Modifying Materials Interactions and Adhesion Values).	Turns on or off

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

Setting	Description	Range
Tangential Impact Velocity	When enabled, Rocky will collect the mean, standard deviation, skewness, and kurtosis values of the impact relative velocity in the tangential direction resulting from the collisions recorded in different regions of the representative particle of the selected particle set, during an interval between two consecutive output times. This can be useful when analyzing particle breakage or granules de-agglomeration, for example.	Turns on or off
Enable per Group Statistics	When enabled, Rocky will collect all the selected intra-particle statistics (from Duration, Frequency, Normal Impact Velocity, Tangential Impact Velocity, Intensities and Stresses) organized per group. This can be useful for analyzing how different particle sets interact to the one being analyzed. In other words, it enables analyzing the contribution of each particle group for the intra-particle collected statistics. **Note:** When enabled, a new **Modules** sub-tab is added to the main **Particle** tab on the **Data Editors** when a particle set is selected in the **Data** panel, as shown in Figure 2.196: Particle | Modules sub-tab available when Enable per Group Statistics is selected (p. 331). In this sub-tab, you can enable or disable the intra-particle statistics collection for the specific particle group you are setting up by marking or clearing the **Enable For This Particle Group** checkbox, respectively. • When marked (default), all the intra-particle selected statistics will be collected for the set (if the set is a supported one), and also its contributions to the other ones. • When cleared, neither the intra-particle selected statistics for the set nor its contributions the other ones will be collected.	Turns on or off

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

Figure 2.196: Particle | Modules sub-tab available when Enable per Group Statistics is selected



What would you like to do next?

- Learn more About the Intra-particle Collision Statistics Module (p. 880)
- Enable and View Collision Statistics for Particle Surfaces (Intra) (p. 346)
- Learn more About Adding and Editing Particle Sets (p. 602)

ABOUT INTER-PARTICLE COLLISION STATISTICS

If you want to expand the set of particle properties available for post-processing, including several statistical properties that may be collected during a simulation, you can choose to collect Inter-particle Collision Statistics prior to processing your simulation.

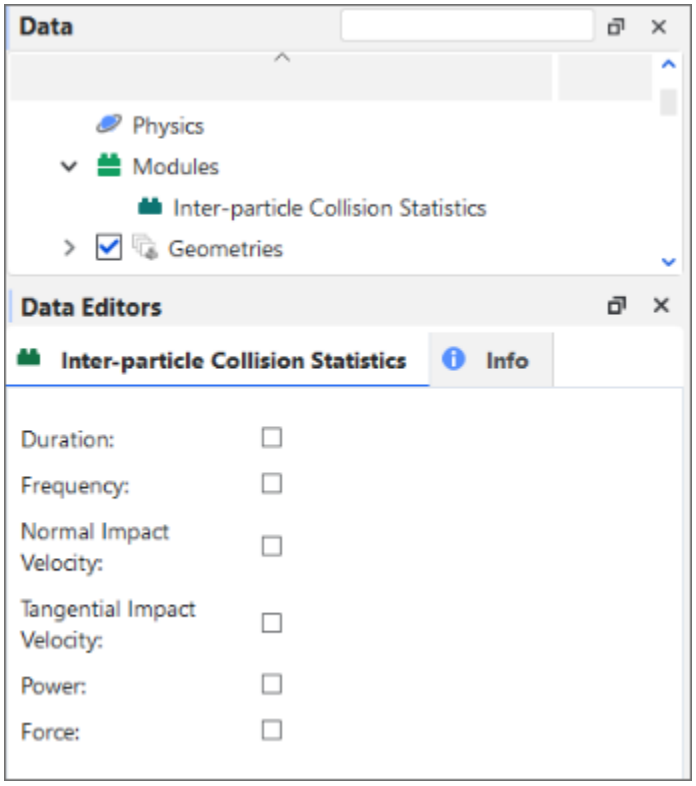
These can be useful when you need to extract data considering all collisions that happened to a certain particle between two output periods. For example, with impact velocity, you could relate that data to the chances of the particle breaking or causing it to de-agglomerate. With duration, you could relate that data to a certain mass or heat transfer process, or to a certain chemical reaction.

About Collecting Inter-particle Collision Statistics

Because collecting collisions statistics can take more processing time, memory, and disk storage due to increased file sizes, Rocky enables you to select from one or more sub-categories of statistics to collect. These are made available through the **Inter-particle Collision Statistics** Module (Figure 1).

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

Figure 2.197: Options in the Data Editors panel when the Inter-particle Collision Statistics Module is enabled



Tip:

To maximize your processing capabilities, choose only the statistics that you require for your analyses.

During processing, Rocky collects the selected collision statistics between two consecutive output time levels for all particles in your project.

Note:

Some Inter-particle Collision Statistics properties will not be provided for Multi-Element (meshed (p. 1392)) particles if the **Meshed Particles Upscaling** feature is enabled. (See also About Meshed Particles Upscaling (p. 1033).)

Tip:

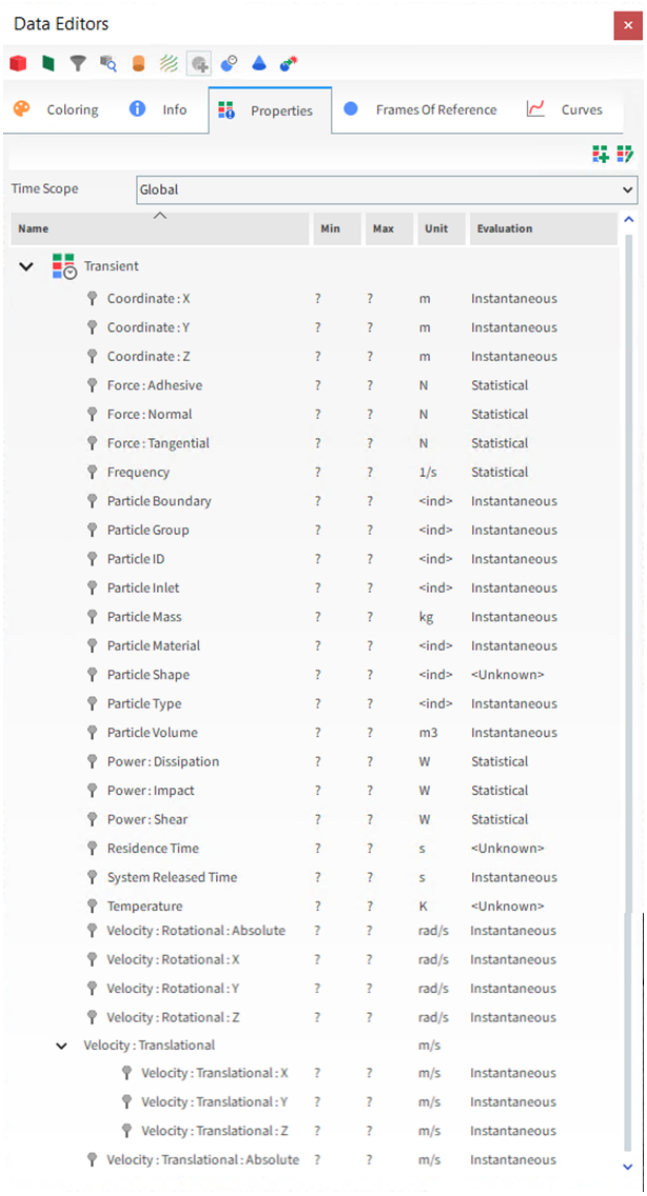
To see a walk-through example of collecting and analyzing Inter-particle collisions statistics, refer to Tutorial - Tablet Coater in the *Rocky Tutorial Guide*.

ABOUT ANALYZING INTER-PARTICLE COLLISION STATISTICS

Once the collisions data is collected (post-processing), specific collisions-related Properties (Figure 2) will be available for the main Particles entity. (See also About Properties (p. 933).) The specific Properties available depends upon which statistics you enabled prior to processing.

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

Figure 2.198: Properties available for the main Particles entity when Inter-particle Collision Statistics is enabled



Tip:

These properties will be categorized as **Statistical** in the **Evaluation** column. (See also About Viewing an Individual Statistic (p. 1098).)

You can then choose to analyze the resulting Properties in a plot or histogram window. (See also Graphing (Plot or Histogram) a Data Set Within Rocky (p. 1101).)

Or, you can choose to display the Properties information in a 3D View window. (See also About 3D View Windows (p. 1040).)

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

By using the **Properties** tab for the main Particles entity, the options on the **Coloring** tab, and/or by using the slider on the Time toolbar (see also About the Time Toolbar (p. 1279)), you can change how the data appears in the window.

Tip:

Learn more by referring to the "Collision statistics" section in the DEM Technical Manual. (From the Rocky **Help** menu, point to **Manuals**, and then click **DEM Technical Manual**.)

Inter-particle Collision Statistics Collection Options

Use Figure 1 above and the table below to understand how to collect Inter-particle Collision Statistics.

Table 1: Modules, Inter-particle Collision Statistics parameter definitions

Setting	Description	Range
Duration	When enabled, Rocky will collect the mean, standard deviation, skewness, and kurtosis values of the duration of the collisions recorded for each individual whole particle or fragment, during an output . This can be useful when you are able to relate the duration to a certain process, such as in simulations involving chemical reactions and/or heat transfer. For example, when a homogeneous system is desired, you might seek similar collision durations across particles to help ensure a uniform distribution of the transferred quantity. **Note:** To make use of this property with Multi-Element (meshed) particles, the **Meshed Particles Upscaling** feature must be disabled. (See also About Meshed Particles Upscaling.)	Turns on or off
Force	When enabled, Rocky will collect the sum of the forces in both the Normal and Tangential directions and for adhesion (if applicable; see note) as recorded for each individual whole particle or fragment. **Note:** Adhesive forces are collected only when an **Adhesive Force** model (other than **None**) is enabled (see also About Physics Parameters), and an Adhesive **Force Fraction** for a particle-boundary and/or particle-particle Materials Interaction pair has a value set higher than zero (see also About Modifying Materials Interactions and Adhesion Values).	Turns on or off
Frequency	When enabled, Rocky will collect the average collision frequency recorded for each individual whole particle or fragment, during an output .	Turns on or off

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

Setting	Description	Range
Normal Impact Velocity	When enabled, Rocky will collect the mean, standard deviation, skewness, and kurtosis values of the impact relative velocity in the normal direction resulting from the collisions recorded for each individual whole particle or fragment, during an output . **Note:** To make use of this property with Multi-Element (meshed) particles, the **Meshed Particles Upscaling** feature must be disabled. (See also About Meshed Particles Upscaling.)	Turns on or off
Power	When enabled, Rocky will collect the dissipation and impact power values resulting from the collisions recorded for each individual whole particle or fragment, during an output .	Turns on or off
Tangential Impact Velocity	When enabled, Rocky will collect the mean, standard deviation, skewness, and kurtosis values of the impact relative velocity in the tangential direction resulting from the collisions recorded for each individual whole particle or fragment, during an output . **Note:** To make use of this property with Multi-Element (meshed) particles, the **Meshed Particles Upscaling** feature must be disabled. (See also About Meshed Particles Upscaling.)	Turns on or off

What would you like to do next?

- Learn more About the Inter-particle Collision Statistics Module (p. 878)
- Enable and View Collision Statistics for Particles Between Other Particles (Inter) (p. 348)
- Learn more About Adding and Editing Particle Sets (p. 602)
- Learn more About Collecting Data in Rocky (p. 306)

ABOUT FLUID-RELATED STATISTICS FOR PARTICLES

For most CFD Coupling options where the fluid flow affects the motion of the particles (1-Way Constant (p. 763), 1-Way Fluent (p. 775), and 2-Way Fluent (p. 795)), you can choose to collect CFD Coupling Particle Statistics prior to processing your simulation.

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

These can be useful when you need to extract data considering the fluid effects upon a particle between two output periods.

Important:

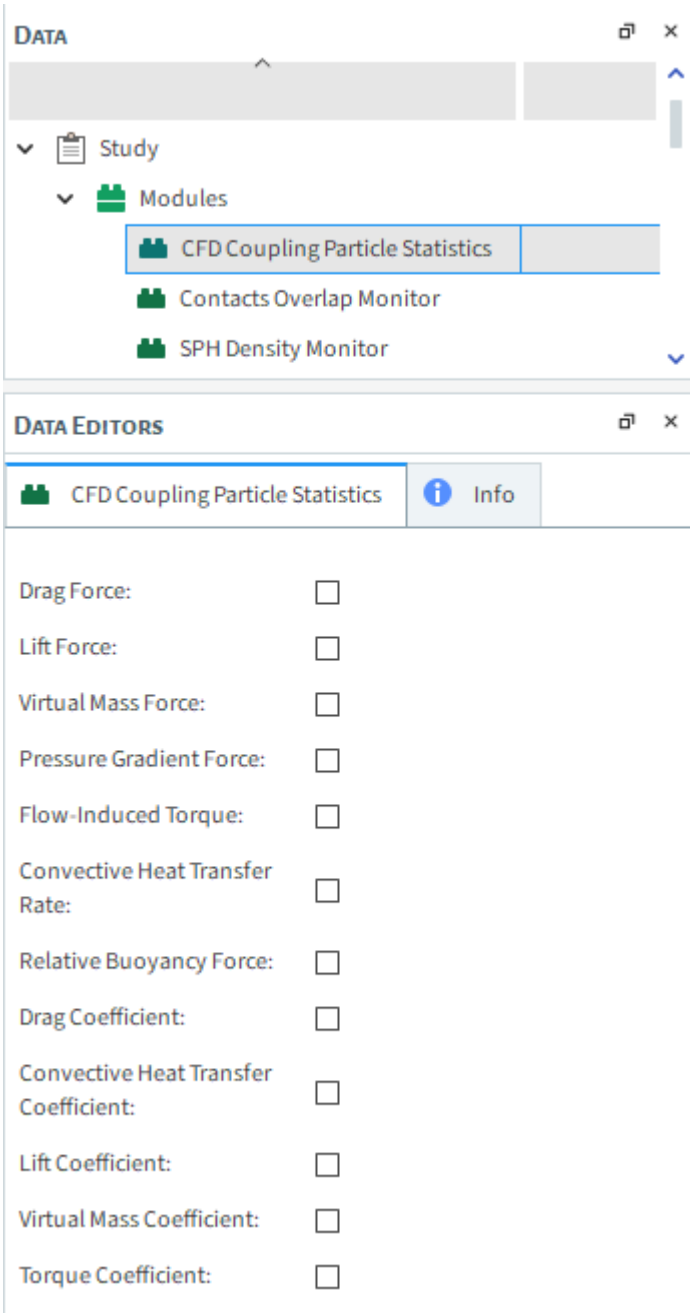
The 2-Way Fluent Semi-Resolved (p. 831) method does not compute the forces individually. Therefore, when this module is enabled within a 2-Way Fluent Semi-Resolved case, it will not return the fluid forces.

About Collecting Fluid-Related Statistics for Particles

Because collecting statistics can take more processing time, memory, and disk storage due to increased file sizes, Rocky enables you to select from one or more sub-categories of statistics to collect. These are made available through the CFD Coupling Particle Statistics Module (Figure 2.199: Options in the Data Editors panel when the CFD Coupling Particle Statistics Module is enabled (p. 337)).

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

Figure 2.199: Options in the Data Editors panel when the CFD Coupling Particle Statistics Module is enabled



Tip:

To maximize your processing capabilities, choose only the statistics that you require for your analyses.

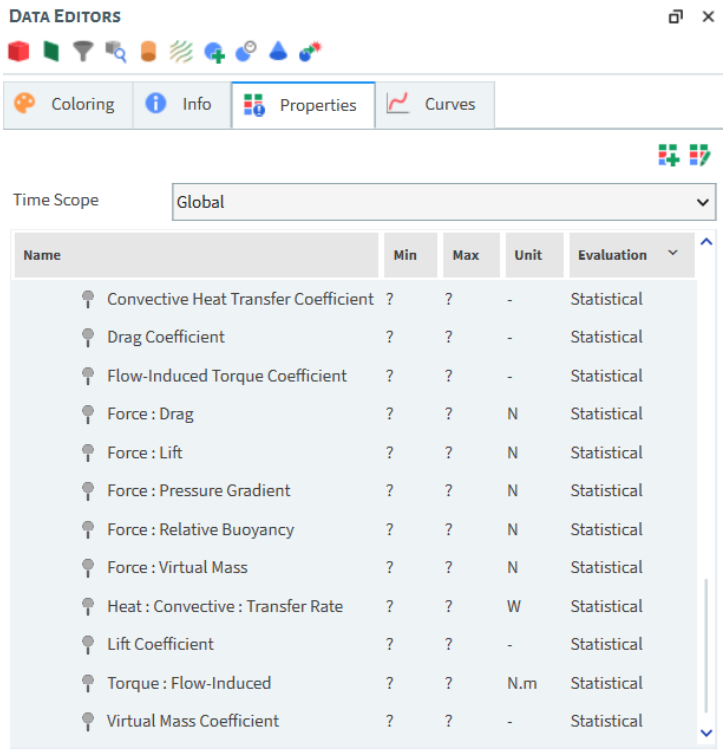
During processing, Rocky collects the selected fluid-related statistics between two consecutive output time levels for all particles in your project.

About Analyzing Fluid-Related Statistics for Particles

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

Once the data is collected (post-processing), specific fluid-related Properties (Figure 2.200: Properties available for the main Particles entity when CFD Coupling Particle Statistics is enabled (p. 338)) will be available for the main Particles entity. (See also About Properties (p. 933).) The specific Properties available depend upon which statistics you enabled prior to processing.

Figure 2.200: Properties available for the main Particles entity when CFD Coupling Particle Statistics is enabled



Tip:

These properties will be categorized as **Statistical** in the **Evaluation** column. (See also About Viewing an Individual Statistic (p. 1098).)

You can then choose to analyze the resulting Properties in a plot or histogram window. (See also Graphing (Plot or Histogram) a Data Set Within Rocky (p. 1101).)

Or, you can choose to display the Properties information in a 3D View window. (See also About 3D View Windows (p. 1040).)

By using the Properties tab for the main Particles entity, the options on the **Coloring** tab, and/or by using the slider on the Time toolbar (see also About the Time Toolbar (p. 1279)), you can change how the data appears in the window.

CFD Coupling Particle Statistics Collection Options

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

Use the Figure 2.200: Properties available for the main Particles entity when CFD Coupling Particle Statistics is enabled (p. 338) image and the Table 2.1: CFD Coupling Particle Statistics Collection Options (p. 339) table below to understand how to collect fluid-related statistics for particles.

Table 2.1: CFD Coupling Particle Statistics Collection Options

Setting	Description	Range
Drag Force	When enabled, Rocky will collect the average drag force applied by fluids as recorded for each individual whole particle or fragment. **Note:** Data is collected only when a **Drag Law** has been defined for the fluid-particle interactions.	Turns on or off
Lift Force	When enabled, Rocky will collect the average lift force applied by fluids as recorded for each individual whole particle or fragment. **Note:** Data is collected only when a **Lift Law** has been defined for the fluid-particle interactions.	Turns on or off
Virtual Mass Force	When enabled, Rocky will collect the average virtual mass force applied by fluids as recorded for each individual whole particle or fragment. **Note:** Data is collected only when a **Virual Mass Law** has been defined for the fluid-particle interactions.	Turns on or off
Pressure Gradient Force	When enabled, Rocky will collect the average pressure gradient force applied by fluids as recorded for each individual whole particle or fragment.	Turns on or off
Flow-Induced Torque	When enabled, Rocky will collect the average torque induced by fluid flow as recorded for each individual whole particle or fragment. **Note:** Data is collected only when a **Torque Law** has been defined for the fluid-particle interactions.	Turns on or off

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

Setting	Description	Range
Convective Heat Transfer Rate	When enabled, Rocky will collect the average convective heat transfer rate exchanged between a particle and the surrounding fluids as recorded for each individual whole particle or fragment. **Note:** Data is collected only when the **Thermal Model** is enabled and a **Convective Heat Transfer Law** has been defined for the fluid-particle interactions.	Turns on or off
Relative Buoyancy Force	When enabled, Rocky will collect the average relative buoyancy force applied by fluids as recorded for each individual whole particle or fragment.	Turns on or off
Drag Coefficient	Enables the collection of the drag coefficient that is used to calculate the drag force exerted over particles in the simulation. **Note:** Drag coefficients are calculated differently according to the **Drag Law** that has been defined for fluid-particle interactions.	Turns on or off
Convective Heat Transfer Coefficient	Enables the collection of the convective heat transfer coefficient that is used to calculate the heat transfer rate between particles and fluid. **Note:** Drag coefficients are calculated differently according to the **Drag Law** that has been defined for fluid-particle interactions.	
Lift Coefficient	Enables the collection of the lift coefficient that is used to calculate the lift force exerted over particles in the simulation. **Note:** Lift coefficients are calculated differently according to the **Lift Law** that has been defined for fluid-particle interactions.	Turns on or off

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

Setting	Description	Range
Virtual Mass Coefficient	Enables the collection of the virtual mass coefficient that is used to calculate the virtual mass force exerted over particles in the simulation. **Note:** Virtual mass coefficients are calculated differently according to the **Virtual Mass Law** that has been defined for fluid-particle interactions.	Turns on or off
Torque Coefficient	Enables the collection of the torque coefficient that is used to calculate the flow-induced torque exerted over particles in the simulation. **Note:** Torque coefficients are calculated differently according to the **Torque Law** that has been defined for fluid-particle interactions.	Turns on or off

What would you like to do next?

- Learn more About the CFD Coupling Particle Statistics Module (p. 873)
- Enable and View Fluid-Related Statistics for Particles (p. 349)
- Learn more About Adding and Editing Particle Sets (p. 602)
- Learn more About Collecting Data in Rocky (p. 306)

ABOUT PARTICLE INSTANTANEOUS ENERGIES

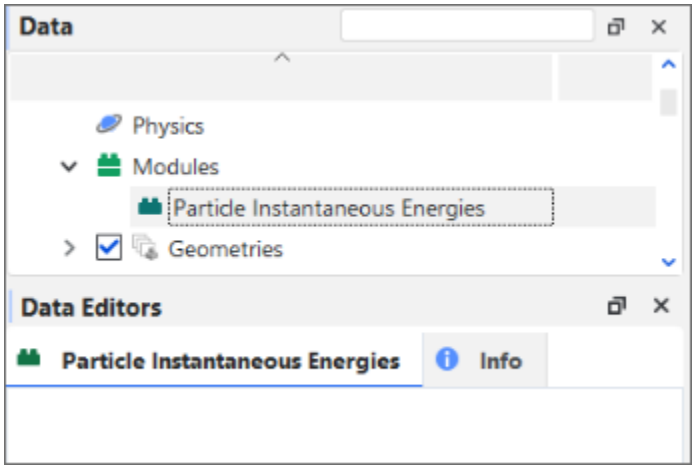
If you want to perform global or partial energy balances in a simulation, you can choose to collect particle instantaneous energies prior to processing your simulation, which enables the calculation of the kinetic and potential energies of each individual particle in the simulation.

About Collecting Particle Instantaneous Energies

You choose to collect these kind of particle energies by enabling the **Particle Instantaneous Energies** Module (Figure 1).

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

Figure 2.201: There are no options in the Data Editors panel when the Particle Instantaneous Energies Module is enabled



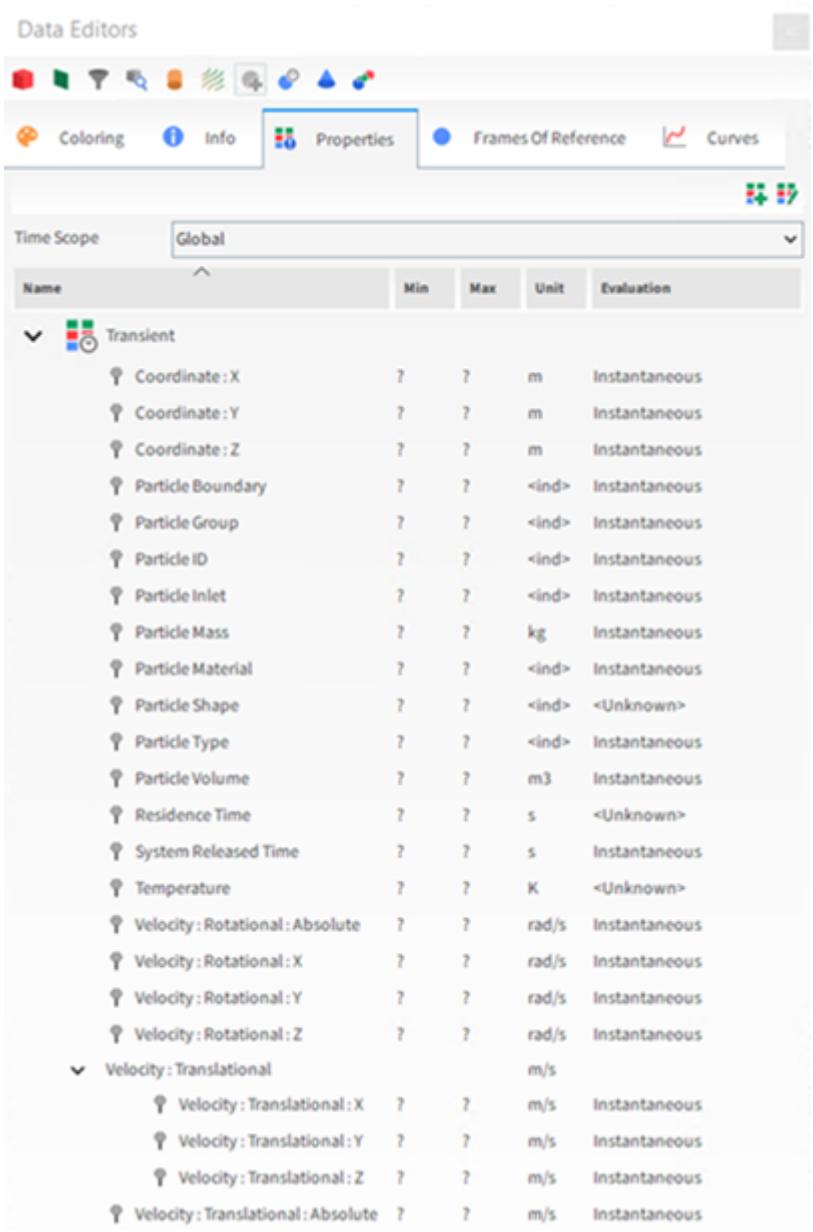
During processing, Rocky collects particle energies data for each particle based upon its translational/rotational velocity and position in space.

About Analyzing Particle Instantaneous Energies

Once the energies data is collected (post-processing), specific energy-related Properties (Figure 2) and Curves (Figure 3) will be available for the main Particles entity. (See also About Properties (p. 933) and About Curves (p. 989).)

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

Figure 2.202: Properties for Particles (simulation-wide) when Particle Instantaneous Energies is enabled



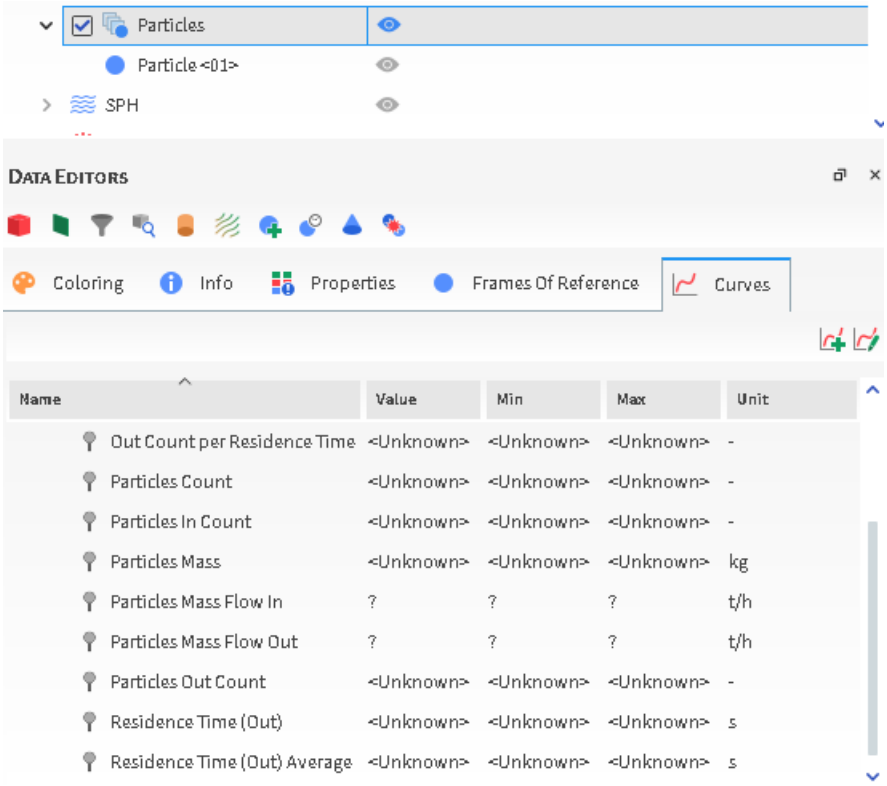
These Properties are based on the following energies calculations:

- **Translational kinetic energy**: The energy related to the rectilinear motion of a particle. It is given by: $E^{k,t}=\frac{1}{2}m_p\mathrm{v}_p^2$ where: m_p is the particle's mass, and v_p is the velocity of its center of mass.
- **Rotational kinetic energy**: This is the energy due to the rotation of a particle around an axis through its center of mass. It is given by: $E^{k,r}=\frac{1}{2}\omega_p^T I\omega_p$ where: ω_p is the particle's angular velocity, and I is the moment of inertia tensor.
- **Potential energy**: This is the energy of a particle due to its position relative to the Earth's gravitational field. In Rocky it is calculated as: $E^p=m_p g\cdot \mathrm{r}_p$ where: m_p is the particle's mass, g is the acceleration due to gravity, and r_p is the position vector of the particle's center of mass. Conventionally, a zero

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

potential energy is attributed to a plane orthogonal to the gravity direction, passing through the origin of the coordinate system.

Figure 2.203: Curves for Particles (simulation-wide) when Particle Instantaneous Energies is enabled

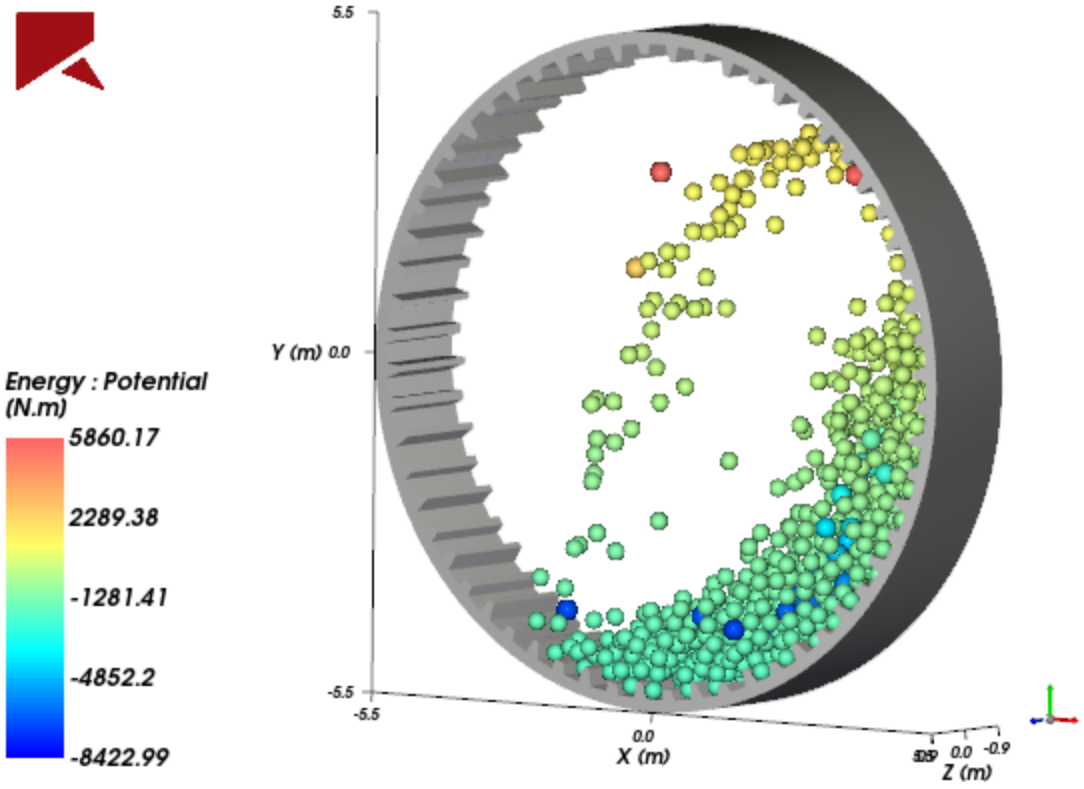


The **Energy Delta** Curve represents a variation (delta) in total energy of the particles recorded during an interval between two consecutive output times.

You can choose to visualize the resulting Properties and Curves in a 3D View window (Figure 3), or analyze the results in a time or other plot window. (See also Graphing (Plot or Histogram) a Data Set Within Rocky (p. 1101).)

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

Figure 2.204: Energy : Potential property for Particles shown in a 3D View window



What would you like to do next?

- Enable and View Instantaneous Energies for Particles (p. 348)
- Learn more About the Particle Instantaneous Energies Module (p. 885)
- Learn more About Adding and Editing Particle Sets (p. 602)
- Learn more About Collecting Data in Rocky (p. 306)

See Also:

- About Properties (p. 933)
- About Curves (p. 989)

ENABLE AND VIEW INTER-GROUP COLLISION STATISTICS FOR PARTICLES

Analyzing Inter-group Collision Statistics for each particle-particle or particle-boundary pair group (see also About Inter-group Collision Statistics (p. 319)) involves turning on the collection of the collision properties you want prior to processing your simulation, and then determining which particle groups and geometries you want to involve in that collection.

After processing, you are then able to analyze the resulting Curves.

To Collect and View Inter-group Collision Statistics:

1. Set up the simulation as you normally would. (See also Set Simulation Parameters (p. 414).)

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

2. Before processing your simulation, do all of the following:

    a. From the **Data** panel, select **Modules**, and then from the **Data Editors** panel, select the **Inter-group Collision Statistics** checkbox.

    b. From the **Data** panel, under **Modules**, select the new **Inter-group Collision Statistics** entry.

    c. From the **Data Editors** panel, select the checkbox for each type of statistics you want collected. (See also About Inter-group Collision Statistics (p. 319).)

    d. For each geometry component that you do *not* want included in the collection of Inter-group Collision Statistics, do the following:

        i. From the **Data** panel, under **Geometries** select the geometry that you want to exclude from Inter-group Collision Statistics collections.

        ii. From the **Data Editors** panel, select the **Geometry | Modules** tab and then under **Inter-group Collision Statistics** clear the **Enable For This Geometry** checkbox.

    e. For each particle group that you do *not* want included in the collection of Inter-group Collision Statistics, do the following:

        i. From the **Data** panel, under **Particles** select the particle group that you want to exclude from Inter-group Collision Statistics collections.

        ii. From the **Data Editors** panel, select the **Particle | Modules** tab and then under **Inter-group Collision Statistics** clear the **Enable For This Particle Group** checkbox.

3. Process the simulation as you normally would. (See also Processing a Simulation (p. 907).)

4. From the **Data** panel, select the main **Particles** entity.

5. From the **Data Editors** panel, select the **Curves** tab.

6. Create a time or cross plot. (See also Graph Data within Rocky by Creating a Plot or Histogram (p. 1124).)

See Also:

- About Inter-group Collision Statistics (p. 319)
- About Collecting Data in Rocky (p. 306)
- About Curves (p. 989)

ENABLE AND VIEW COLLISION STATISTICS FOR PARTICLE SURFACES (INTRA)

Analyzing collision statistics for particle surfaces involves first turning on the collection of Intra-particle Collision Statistics prior to processing the simulation (see also About Intra-Particle Collision Statistics (p. 324)), and then viewing the resulting Properties for a supported Particle set (p. 1393) on the surface of a representative particle in a **Particles Details** window.

By using the slider on the **Time** toolbar (see also About the Time Toolbar (p. 1279)), you can see how the data changes at different points in time.

To Collect and View Collision Statistics for Particle Surfaces (Intra:

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

1. Set up your simulation as you normally would (see also Setting Up a Simulation (p. 411)). Note: If you decide to enable the Thermal Model, know that while the Temperature property will not be available to view on the **Particles Details** window, it can still be viewed from the main **Particles** entity or through a **3D View** window. (See also Enable Thermal Modeling Calculations (p. 400).)

2. Before processing your simulation, do all of the following:

   a. From the **Data** panel, select **Modules**, and then from the **Data Editors** panel, select the **Intra-particle Collision Statistics** checkbox.

   b. From the **Data** panel, under **Modules**, select the new **Intra-particle Collision Statistics** entity.

   c. From the **Data Editors** panel, enable the checkboxes for the type of statistics you want collected. (See also About Intra-Particle Collision Statistics (p. 324).)

   d. Ensure you have added at least one Particle set (p. 1393) that is eligible for intra-particle collision statistics collection. Note: Intra-particle collision statistics can be collected only on single sized particle sets that have no breakage enabled and that make use of the following shapes:

      - Polyhedron (rigid or flexible)
      - Custom Polyhedron (convex or concave, rigid or flexible)
      - Straight Fiber or Custom Fiber (flexible only)

      (See also About Adding and Editing Particle Sets (p. 602).)

3. Process your simulation as you normally would (see also About Starting a Simulation (p. 908).)

4. From the **Data** panel, under **Particles**, select the Particle set for which you want to view intra-particle collision statistics, and then click the **View** button. A new **Particles Details** window opens showing a particle that represents the whole Particle set.

5. Display intra-particle collision statistics Properties on the view in one of the following ways:

   - From the **Properties** tab, click a Property and then drag and drop it on the **Particles Details** window. (See also About Properties (p. 933).) Tips: You may also make use of Time Statistics Properties. (See also About Adding and Editing Time Statistics Properties (p. 1023).)
   - From the **Coloring** tab, use the **Property** lists to change what is colored in the **Particles Details** window. (See also Use the Coloring Tab to Change the Preview for a Particle Set (p. 697).)

6. Use the slider on the **Time** toolbar to change what is displaying data. (See also About the Time Toolbar (p. 1279).) Note: The data displayed at a given was collected in the interval between that and the previous one.

Tips:

- Use your mouse to zoom, pan, and rotate the view, just as you would in a 3D View window. (See also Use the Mouse, Keyboard, or Toolbar to Change a 3D View (p. 1047).)
- Change the look of the window by right-clicking the window, or by using the options on the Windows Editors panel. (See also About Using the Window Editors Panel to Change the Selected Particles Details Window (p. 683).)

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

See Also:

- About Collecting Data in Rocky (p. 306)
- About Intra-Particle Collision Statistics (p. 324)
- About Properties (p. 933)

ENABLE AND VIEW COLLISION STATISTICS FOR PARTICLES BETWEEN OTHER PARTOCLES (INTER)

Analyzing inter-particle collision statistics (see also About Inter-particle Collision Statistics (p. 331)) involves turning on the collection of the collision properties you want prior to processing your simulation.

After processing, you are then able to analyze the resulting Properties as you normally would.

To Collect and View Collision Statistics for Particles Between Other Particles (Inter:

1. Set up the simulation as you normally would. (See also Set Simulation Parameters (p. 414).)
2. Before processing your simulation, do all of the following:
   - From the **Data** panel, select **Modules**, and then from the **Data Editors** panel, enable the **Inter-particle Collision Statistics** checkbox.
   - From the **Data** panel, under **Modules**, select the new **Inter-particle Collision Statistics** entry, and then from the **Data Editors** panel, enable the checkboxes for the type of statistics you want collected. (See also About Inter-particle Collision Statistics (p. 331).)
3. Process the simulation as you normally would. (See also Processing a Simulation (p. 907).)
4. From the **Data** panel, select the main **Particles** entity.
5. From the **Data Editors** panel, select the **Properties** tab.
6. Create a plot (see also Graph Data within Rocky by Creating a Plot or Histogram (p. 1124)), or visualize the data in a 3D View window (see also About 3D View Windows (p. 1040)).

See Also:

- About Inter-particle Collision Statistics (p. 331)
- About Collecting Data in Rocky (p. 306)
- About Properties (p. 933)

ENABLE AND VIEW INSTANTANEOUS ENERGIES FOR PARTICLES

Analyzing Particle Instantaneous Energies (see also About Particle Instantaneous Energies (p. 341)) involves turning on the collection of energies data prior to processing your simulation.

After processing, you are then able to analyze the resulting Properties as you normally would.

To Collect and View Inter-group Collision Statistics:

1. Set up the simulation as you normally would. (See also Set Simulation Parameters (p. 414).)

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

2. Before processing your simulation, do all of the following:
   - From the **Data** panel, select **Modules**.
   - From the **Data Editors** panel, select the **Particle Instantaneous Energies** checkbox.
3. Process the simulation as you normally would. (See also Processing a Simulation (p. 907).)
4. From the **Data** panel, select the main **Particles** entity.
5. From the **Data Editors** panel, select the **Properties** tab.
6. Create a 3D View (see also View Geometries, Particles, Points, and Fluids in 3D (p. 1041)) or plot (see also Graph Data within Rocky by Creating a Plot or Histogram (p. 1124)).

See Also:

- About Particle Instantaneous Energies (p. 341)
- About Collecting Data in Rocky (p. 306)
- About Properties (p. 933)

ENABLE AND VIEW FLUID-RELATED STATISTICS FOR PARTICLES

Analyzing fluid-related statistics for particles (see also About Fluid-Related Statistics for Particles (p. 335)) involves turning on the collection of the fluid-related properties you want prior to processing your CFD Coupling simulation.

After processing, you are then able to analyze the resulting Properties for the main Particles entity as you normally would.

To Collect and View Fluid-Related Statistics for Particles

1. Set up your CFD Coupling simulation as you normally would. (See also Set Simulation Parameters (p. 414).)
2. Before processing your simulation, do all of the following:
   - From the **Data** panel, select **Modules**, and then from the **Data Editors** panel, enable the **CFD Coupling Particle Statistics** checkbox.
   - From the **Data** panel, under **Modules**, select the new **CFD Coupling Particle Statistics** entry, and then from the **Data Editors** panel, enable the checkboxes for the type of statistics you want collected. (See also About Fluid-Related Statistics for Particles (p. 335).)
3. Process the simulation as you normally would. (See also Processing a Simulation (p. 907).)
4. From the **Data** panel, select the main **Particles** entity.
5. From the **Data Editors** panel, select the **Properties** tab.
6. Create a plot (see also Graph Data within Rocky by Creating a Plot or Histogram (p. 1124)), or visualize the data in a 3D View window (see also About 3D View Windows (p. 1040)).

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

See Also:

- About Fluid-Related Statistics for Particles (p. 335)
- About Collecting Data in Rocky (p. 306)
- About Properties (p. 933)

COLLECTING DATA ON CONTACTS AND ENERGY SPECTRA

In Rocky, contacts and energy spectra data is collected only if you first opt in to collecting them before you process your simulation. After choosing to collect it, contacts data is always collected right from the start of the simulation. However, data for energy spectra waits to be collected until passing certain thresholds that you define.

TYPES OF ENERGY SPECTRA

Depending on the source of the energy values considered, two types of energy spectra plots are available in Rocky: **Contacts** Energy Spectra and **Particles** Energy Spectra.

- With **Contacts Energy Spectra**, the energy values are collected collision-wise during the simulation and the resulting data is categorized by the contact pair (particle group and/or geometry).
- With **Particles Energy Spectra** the energy values collected during the simulation are related only to particles, and the resulting data is classified by size and particle group.

Based upon your energy analysis needs, you can use either or both types of energy spectra in your simulation.

Important:

The energy scale in all energy spectra plots is logarithmic, so zero is not a valid value for the minimum energy, therefore for Contacts and Particles energy spectra the minimum energy values must be greater than zero.

Use the topics below to help you understand what, how, and when contacts and energy spectra data is collected in Rocky.

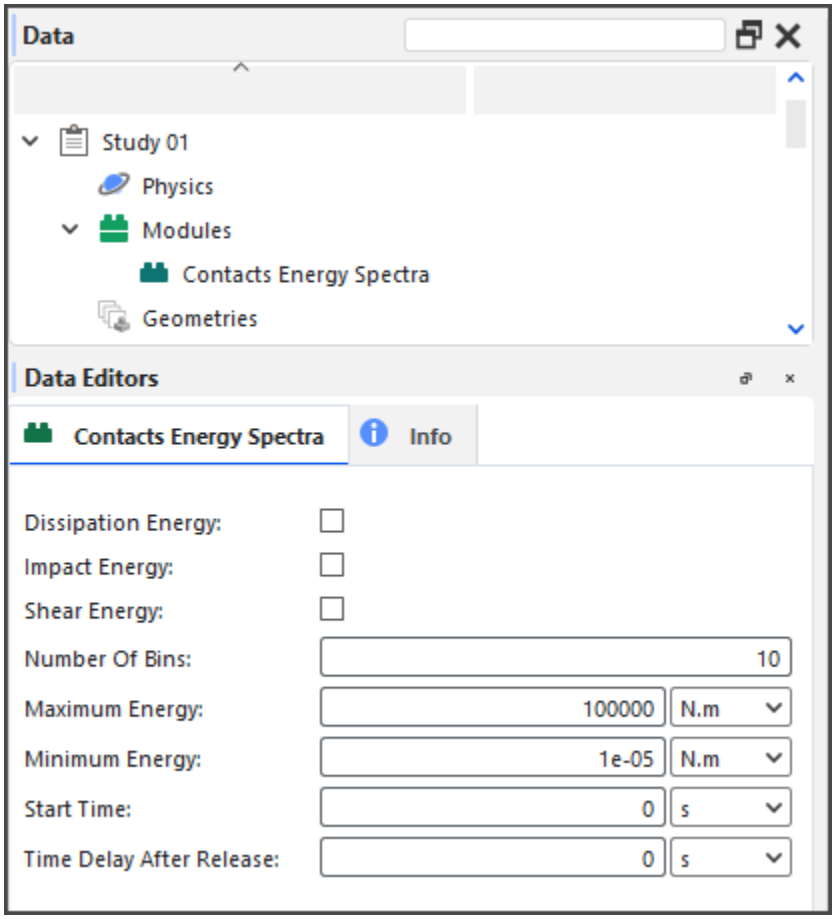
ABOUT CONTACTS ENERGY SPECTRA

In Rocky, it is possible simulate the breakage behavior of particles. (See also Enable Particle Breakage Calculations (p. 397).) However, simulations with breakage enabled can increase the computational cost, taking longer to process the simulation. With this in mind, Rocky has another feature you can use to analyze breakage: Contacts Energy Spectra.

This useful tool collects different kinds of energy statistics per contact pair (particle group and/or geometry) and size, which helps provide insight into the distribution of the energies of all collisions that occurred in a simulation up to a given time. In this version of Rocky, these statistics are collected through the **Contacts Energy Spectra** module (Figure 1).

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

Figure 2.205: Options in the Data Editors panel when the Contacts Energy Spectra Module is enabled



This kind of energy spectra is often used in conjunction with Particles Energy Spectra. (See also About Particles Energy Spectra (p. 358).)

See the sections below for more information about this type of energy statistic, and how it is collected, calculated, and viewed in Rocky.

About Collecting Contact-Based Energy Statistics

For contact-based analysis, the energy statistics of particle collisions are collected per collision; i.e., every separate particle-to-particle or particle-to-geometry impact will have its resulting energy collision stored. Each statistics pair-which is composed of either two Particle sets or one Particle set and one Geometry-will present a separate Energy Spectra curve. In this version of Rocky, only particles and geometries belonging to particle groups and geometry components that have been enabled for energy spectra collection will be recorded (Figures 2 and 3).

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

Figure 2.206: Additional module options for Particle groups when the Contacts Energy Spectra module is enabled

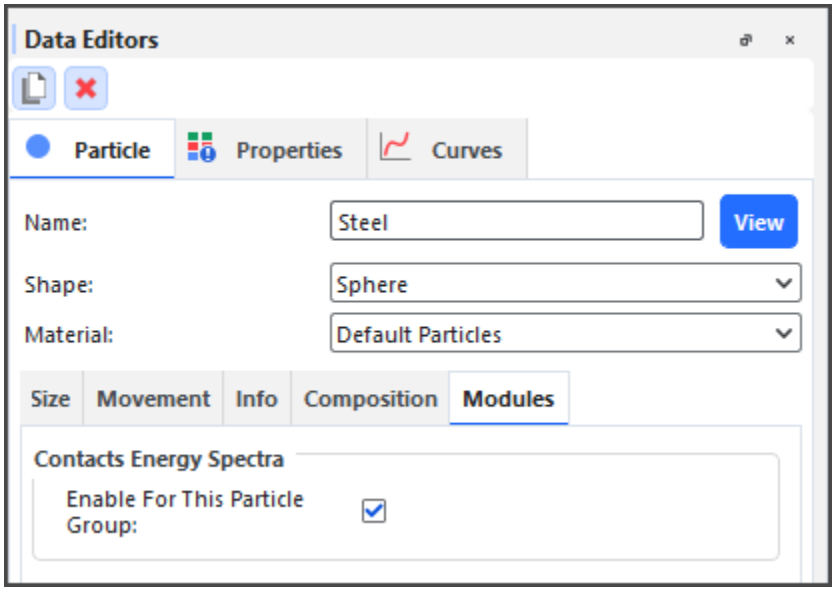
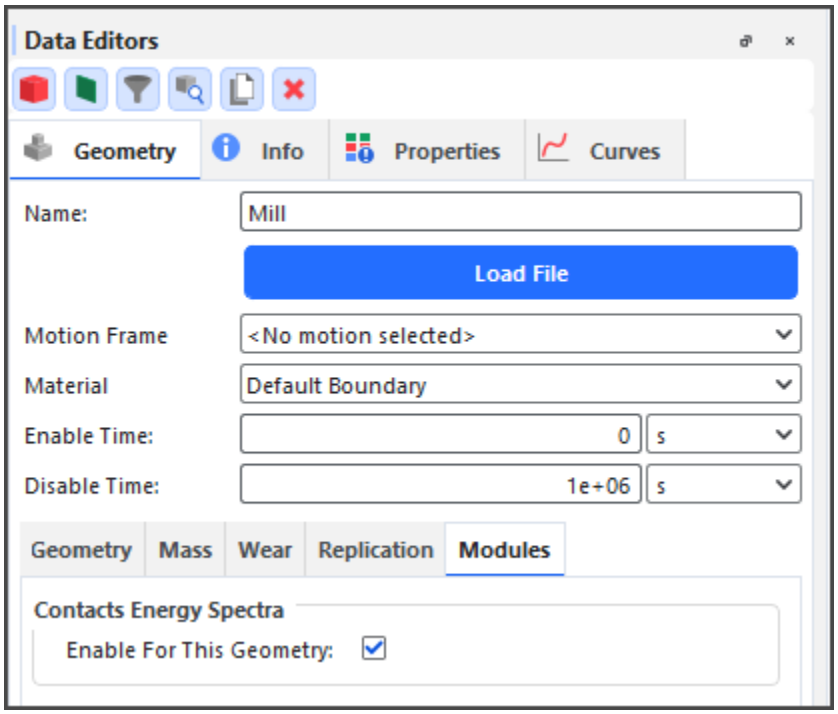


Figure 2.207: Additional module options for a geometry component when the Contacts Energy Spectra module is enabled



These curves are based upon three different types of collision energy, each of which can be useful for different types of analyses: Dissipation Energy, Impact Energy, and Shear Energy.

Because collecting statistics can take more processing time, memory, and disk storage due to increased file sizes, Rocky enables you to select which of these three collision energies you want to collect, and enables you to choose which particle groups and geometry components will participate in energy spectra collection.

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

About Setting Time Delays

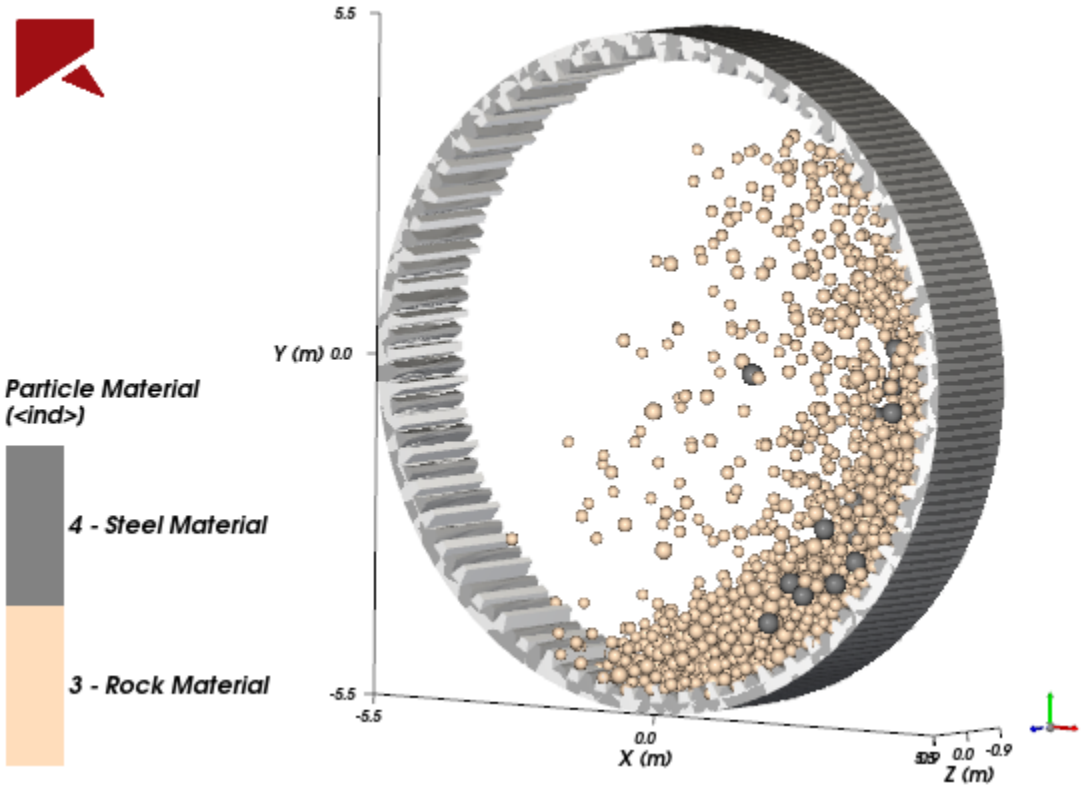
It is important to realize that data is only collected after both the **Start Time** and **Time Delay After Release** values are reached. Data begins being collected at the **Start Time** but only for the particles that were released before the **Time Delay After Release** value. This is to allow enough time for the particle flow within your simulation to reach a steady state before the energy spectra calculations begin.

For example, if the **Start Time** is 5s, and the **Time Delay After Release** is 3s, a particle entering the simulation at 4s will only be accounted for after 7s of simulation time.

Applications for Contacts Energy Analysis

One common application of this type of analysis is for comminution devices like SAG mills (Figure 4), which use steel balls or other kinds of grinding media to help break rock or ore material into smaller fragments.

Figure 2.208: SAG Mill Simulated with both Grinding Media (Steel Balls) and Ore (Rock)



Although mills have great energy consumption, only a small part of this energy is converted into actual particle fragmentation. This inefficiency is mainly due to the fact that not all impacts lead to breakage. Impacts of low energy will not cause breakage, while excessive intensity impacts apply only part of the total energy to the breakage process. The rest of this energy is lost.

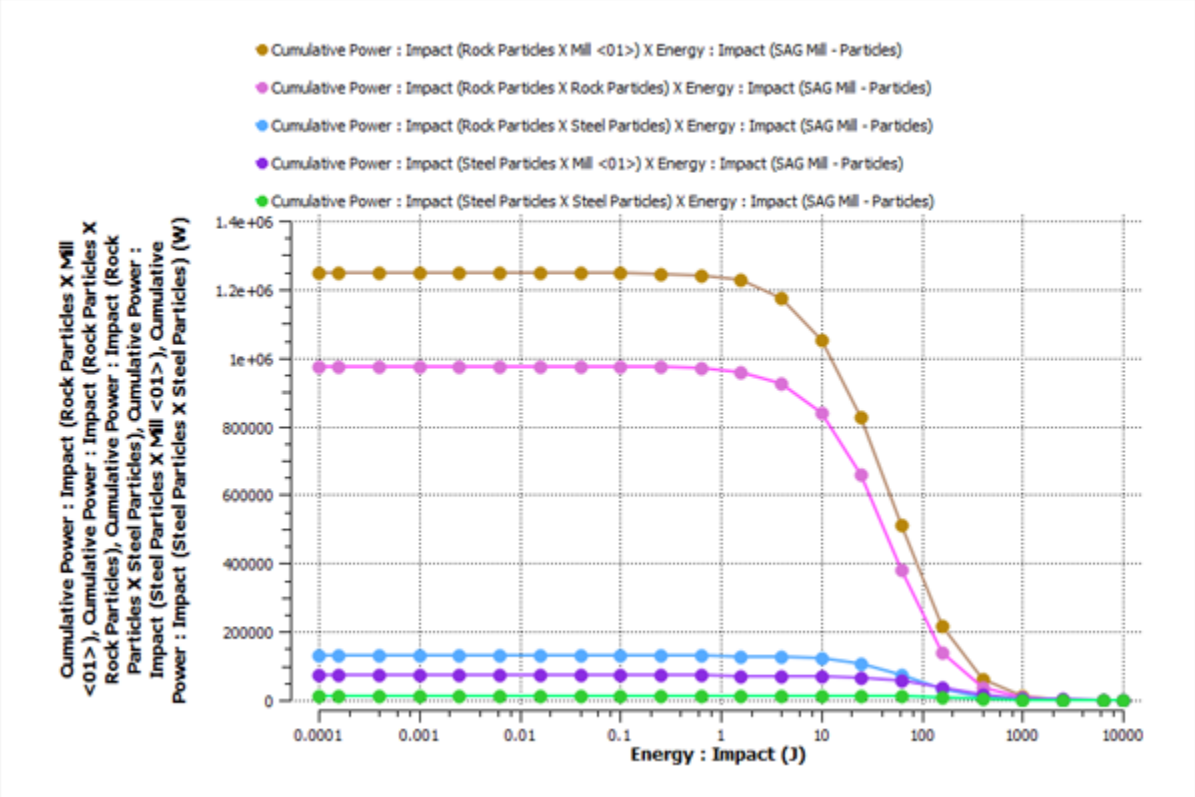
Therefore, it is important to understand how power is consumed during comminution processes in order to improve grinding efficiency. The Contact-based analysis is especially useful when combined with the Cumulative Power analysis for a comminution process because you can get an estimate of how collision energy is distributed amongst contacts.

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

In this way, by comparing contacts energy spectra per material pair for different equipment designs, higher collision energies involving the material to be broken (rock-rock, rock-ball, or rock-wall, for example) might lead to higher particle breakage. On the other hand, collision energies involving only the equipment or grinding materials (ball-ball or ball-wall, for example) could be considered wasted by not leading to particle breakage, and should be minimized through improved designs.

Contacts Energy Spectra curves can be displayed by Power, Cumulative Power (Figure 5), and Rate.

Figure 2.209: Examples of Contacts Impact Energy Spectra curves for various particle-to-particle and particle-to-geometry contact pairs



Tip:

Refer to the DEM Technical Manual for implementation details. (From the **Help** menu, point to **Manuals**, and then click **DEM Technical Manual**.)

Viewing Contacts Energy Spectra

If prior to processing your simulation, you enabled **Contacts Energy Spectra** to be collected, then while the simulation is processing, Rocky calculates the selected energy curves (Dissipation, Impact, and/or Shear) for each pair of particle-to-particle or particle-to-geometry contact types, each generated by the power, cumulative power, and collisions rate (Figure 6).

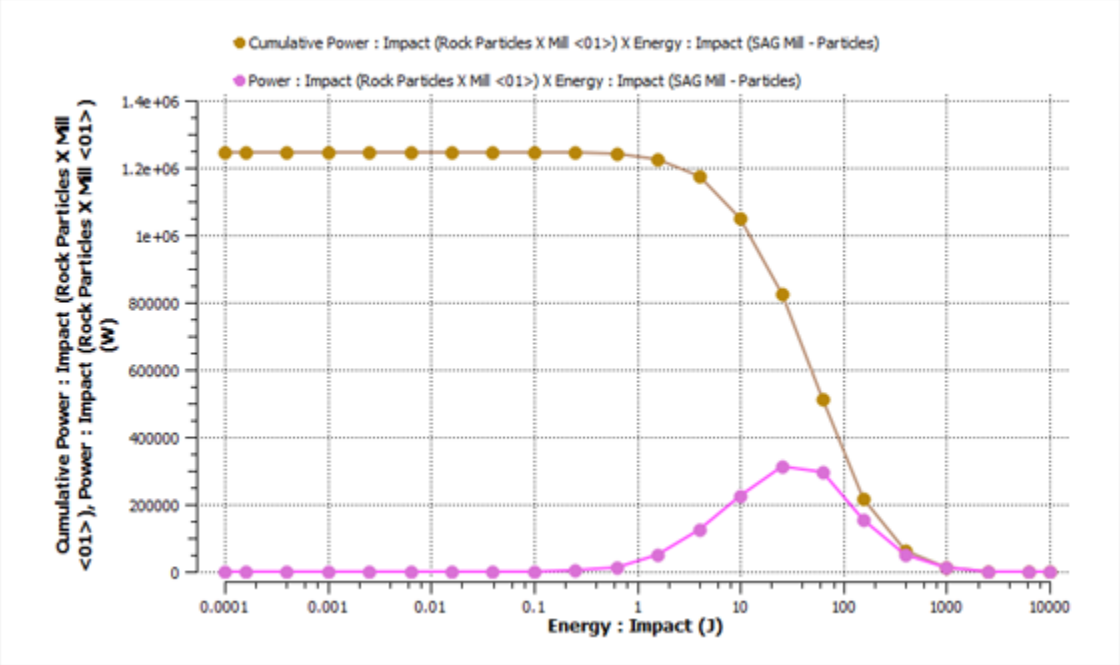
Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

Figure 2.210: Curves tab for Particles showing three sections for Energy

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

You can then create a Cross plot of the Curves you are interested in analyzing (Figure 7).

Figure 2.211: Example cross plot of Contacts Energy Spectra showing both Cumulative Power and Power, for impact collisions between particles and geometries



Tip:

For more information about these Curves, see About Curves (p. 989).

About Opening Older Energy Spectra Projects

When opening projects that were created and processed in versions of Rocky prior to 2022 R1, Rocky will still read the energy spectra data in those projects and will display the energy spectra Curves as usual.

However, if you want to modify and then re-process an older energy spectra simulation in 2022 R1 or later versions, you will need to enable the energy spectra modules and define the type of collection parameters you want prior to processing the simulation.

The same is true for projects that you have created but not processed in older versions of Rocky; you must enable and define the energy spectra modules manually before processing your simulation.

Contacts Energy Spectra Collection Options

Use Figures 1-3 above and the tables below to understand how to collect contact-based energy statistics.

Table 1: Modules, Contacts Energy Spectra parameter definitions

Setting	Description	Range
Dissipation Energy	When enabled, collects contact-based statistics for dissipation energy, which is the fraction of the	Turns on or off

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

Setting	Description	Range
	mechanical energy transformed irreversibly into other forms of energy during a collision.	
Impact Energy	When enabled, collects contact-based statistics for impact energy, which is the maximum energy transferred during a collision. **Tip:** When Rocky simulates an actual breakage process (see also Enable and View Particle Breakage), this is usually the energy type considered to quantify the damage on the particles.	Turns on or off
Shear Energy	When enabled, collects contact-based statistics for shear energy, which is the work done by the tangential contact forces during a collision.	Turns on or off
Number of Bins	This defines how many sub-intervals the specified energy interval is evenly (using a logarithmic scale) subdivided. Determines the the resolution of the resulting Curves. The higher the value, the better the resolution.	Positive values
Maximum Energy	This defines the right bound of the interval of energies that will be subdivided into bins for the construction of the energy spectra Curves.	Positive values
Minimum Energy	This defines the left bound of the interval of energies that will be subdivided into bins for the construction of the energy spectra Curves.	Positive values
Start Time	The amount of time you want to wait before starting to register energy spectra data. **Tip:** It is best practice to set your **Start Time** to begin after a steady state has been reached in your particle flow.	Value must be positive but less than **Simulation Duration**, which is located on the **Solver | Time** sub-tab. (See also About Solver Parameters.)
Time Delay after Release	The amount of time you want to wait after a particle has been released before starting to register energy spectra data.	Value must be positive but less than **Simulation Duration**, which is located on the **Solver | Time** sub-tab. (See also About Solver Parameters.)

Table 2: Individual Geometry, Modules sub-tab parameter definitions

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

Setting	Description	Range
Enable For This Geometry	When enabled, the energy values involving this geometry will be recorded for energy spectra purposes. When cleared, the energy values involving this geometry will be ignored.	Turns on or off

Table 3: Individual Particle Group, Modules sub-tab parameter definitions

Setting	Description	Range
Enable For This Particle Group	When enabled, the energy values involving particles from this group will be recorded for energy spectra purposes. When cleared, the energy values involving particles from this group will be ignored.	Turns on or off

What would you like to do?

- Enable and View Data for Contacts Energy Spectra (p. 367)
- Learn more About Collecting Data (p. 306)
- Learn more About Curves (p. 989)

See Also

- About Particles Energy Spectra (p. 358)
- View Frequently Used Tasks (p. 371)

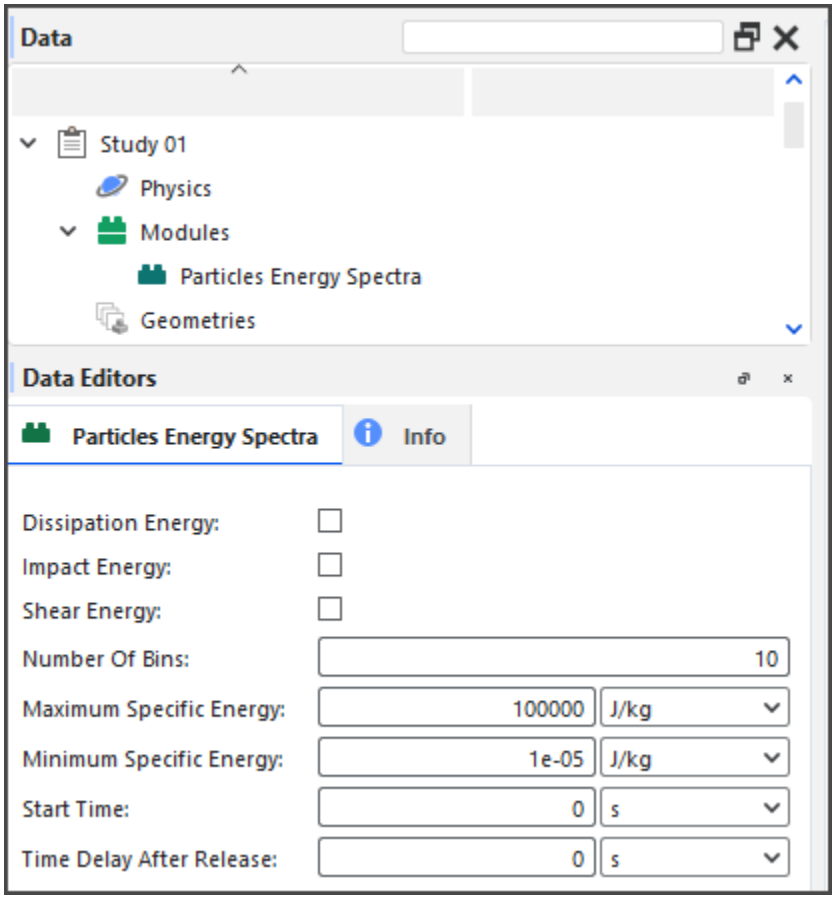
ABOUT PARTICLES ENERGY SPECTRA

In Rocky, it is possible simulate the breakage behavior of particles. (See also Enable Particle Breakage Calculations (p. 397).) However, simulations with breakage enabled can increase the computational cost, taking longer to process the simulation. With this in mind, Rocky has another feature you can use to analyze breakage: Particles Energy Spectra.

This useful tool collects different kinds of energy statistics per particle type and size category defined in its PSD (p. 1394), which can help predict breakage and attrition rates for continuous processes such as grinding mills. In this version of Rocky, these statistics are collected through the **Particles Energy Spectra** module (Figure 1).

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

Figure 2.212: Options in the Data Editors panel when the Particles Energy Spectra Module is enabled



This kind of energy spectra is often used in conjunction with Contacts Energy Spectra. (See also About Contacts Energy Spectra (p. 350).)

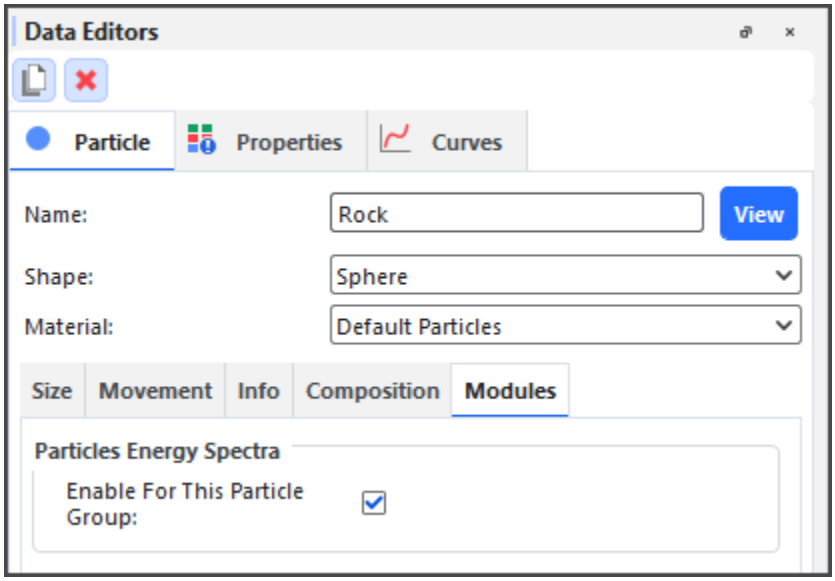
See the sections below for more information about this type of energy statistic, and how it is collected, calculated, and viewed in Rocky.

About Collecting Particles-Based Energy Statistics

For particle-based analysis, relevant energy values associated to the collisions experienced by each individual particle collected during the simulation. In this version of Rocky, only particles belonging to particle groups that have been enabled for energy spectra collection will be recorded (Figure 2).

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

Figure 2.213: Additional module options for Particle groups when the Particles Energy Spectra module is enabled



Rocky then classifies and accumulates the associated specific energy values (energy per particle mass) according to a predefined set of energy levels, and represents the resulting cumulative values as curves.

For Particles Energy Spectra analysis, each particle group and particle size category will present a separate Energy Spectra curve. These curves are based upon three different types of collision energy, each of which can be useful for different types of analyses:

- **Dissipation Energy**, which is the particle's mechanical energy transformed irreversibly into other forms of energy during a collision.
- **Impact Energy**, which is the energy considered in breakage models for quantifying the damage on the particle's material that can lead to breakage. Tips: When Rocky simulates an actual breakage process (see also Enable and View Particle Breakage), this is usually the energy type considered to quantify the damage on the particles.
- **Shear Energy**, which is the energy used to predict the abrasive wear of boundaries.

Because collecting statistics can take more processing time, memory, and disk storage due to increased file sizes, Rocky enables you to select which of these three particles energies you want to collect, and enables you to choose which particle groups will participate in energy spectra collection.

About Setting Time Delays

It is important to realize that data is only collected after both the **Start Time** and **Time Delay After Release** values are reached. Data begins being collected at the **Start Time** but only for the particles that were released before the **Time Delay After Release** value. This is to allow enough time for the particle flow within your simulation to reach a steady state before the energy spectra calculations begin.

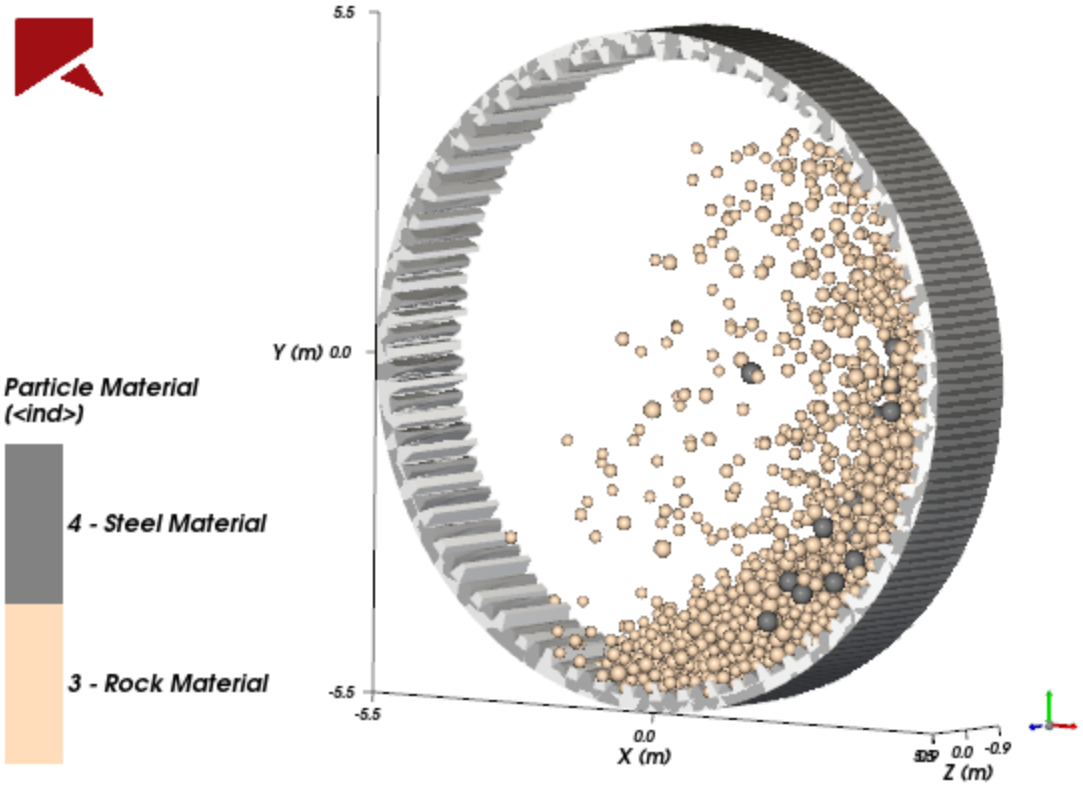
For example, if the **Start Time** is 5s, and the **Time Delay After Release** is 3s, a particle entering the simulation at 4s will only be accounted for after 7s of simulation time.

Applications for Particles Energy Analysis

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

One common application of this type of analysis is for comminution devices like SAG mills (Figure 3), which use steel balls or other kinds of grinding media to help break rock or ore material into smaller fragments.

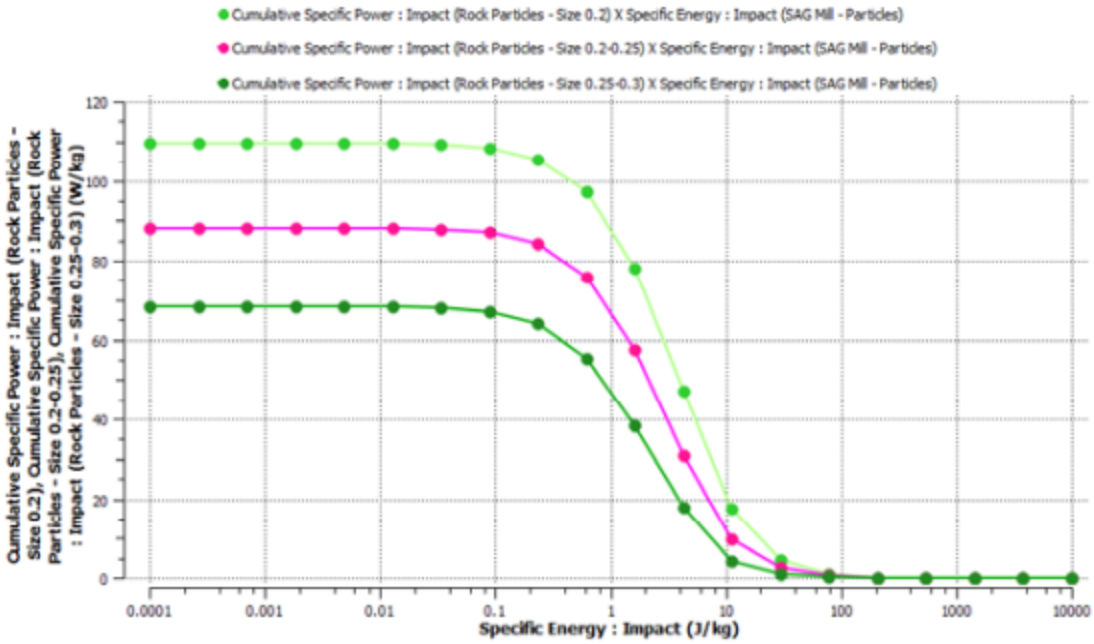
Figure 2.214: SAG Mill Simulated with both Grinding Media (Steel Balls) and Ore (Rock)



By looking at energy levels applied to particles, for instance, you can predict breakage rates.

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

Figure 2.215: Particles Energy Spectra curves related to Impact Energy

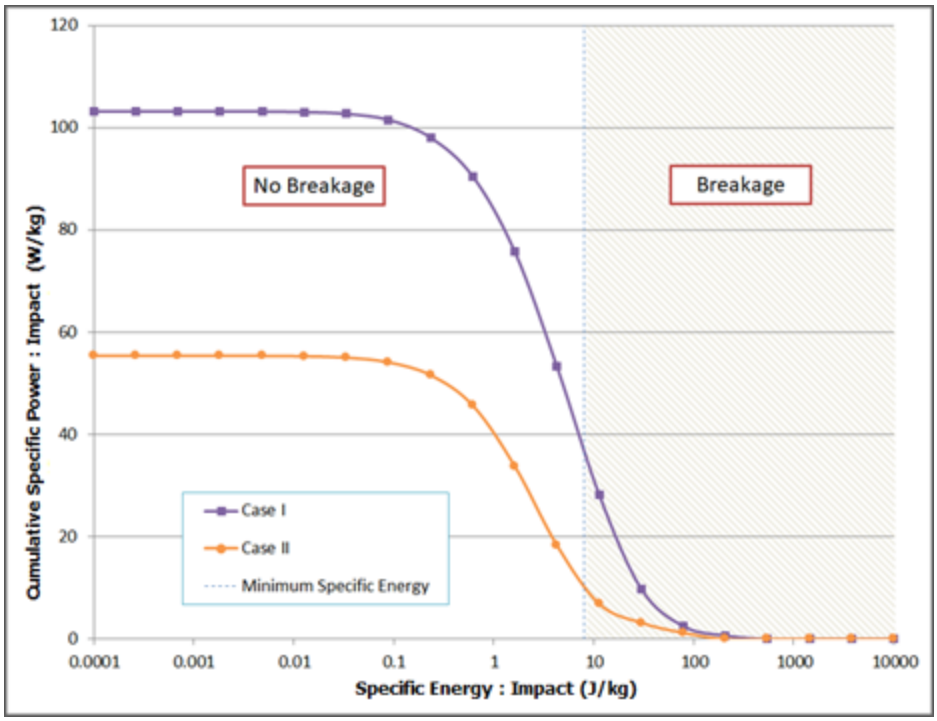


In the example shown in Figure 4 above, the Y-axis gives the **Cumulative Specific Power : Impact** values resulting from all collisions starting from the **Start Time** time you enter until the end of the simulated time (see also About Solver Parameters (p. 464)). The value of the Y-coordinate of a given point on the curve is obtained by dividing the cumulative specific energy by the time elapsed since the referred starting time. This value accounts for all collisions with a specific impact energy equal or higher than the specific energy given by its X-coordinate.

This kind of energy spectra curve is useful for the reason that all collisions with specific impact energy higher than a known threshold value may lead to particle breakage, while the remaining collisions will not, as shown in Figure 5 below. Therefore, the cumulative specific power available for breakage in that case will be given by the Y-coordinate in the curve, corresponding to the breakage threshold value.

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

Figure 2.216: Particles Energy Spectra comparison



Important:

This threshold value is the minimum value of impact energy that can break a particle, which is a known value derived from experiments. It will therefore change depending upon the material being simulated.

Tips:

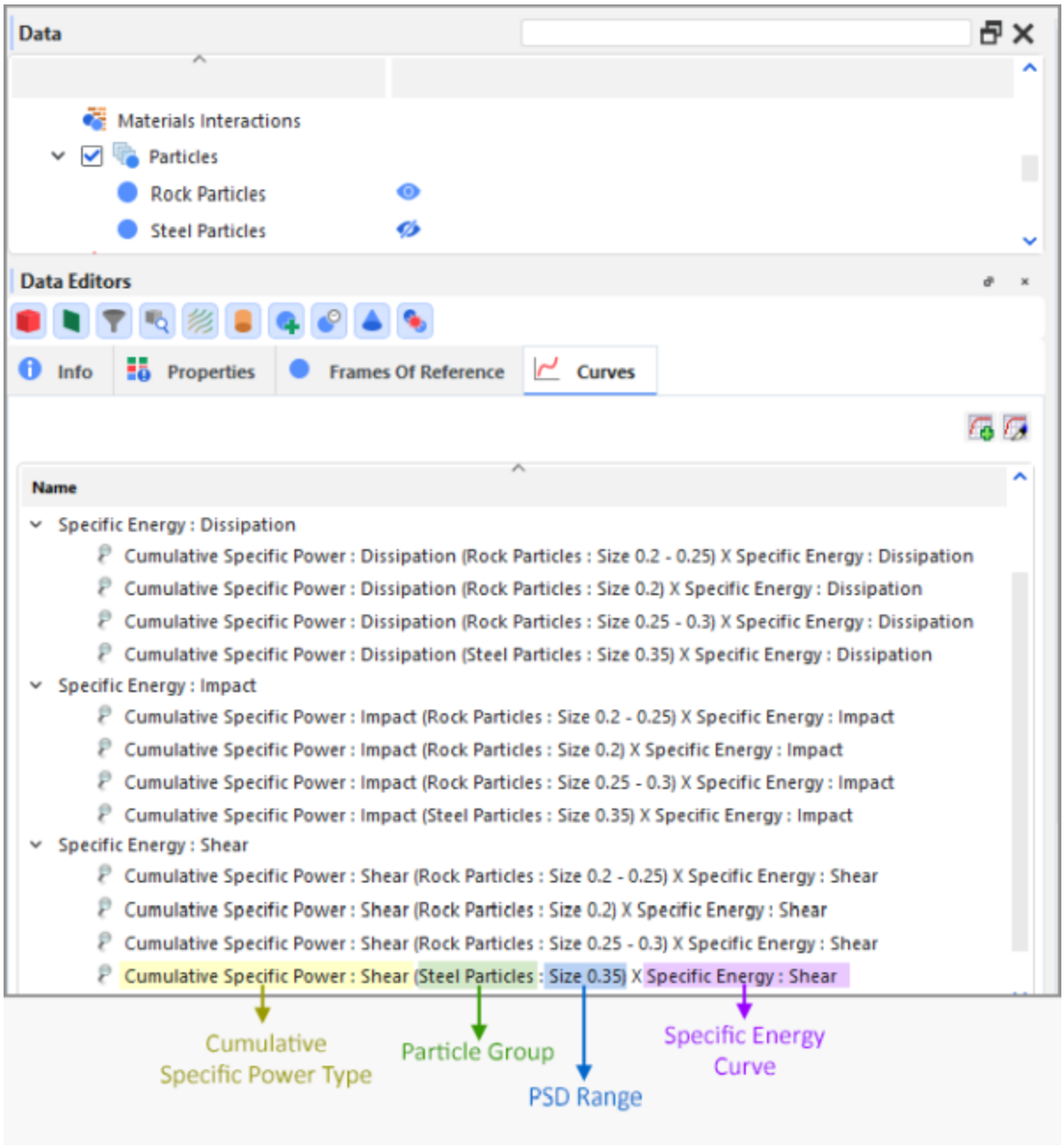
- Refer to the DEM Technical Manual for implementation details. (From the **Help** menu, point to **Manuals**, and then click **DEM Technical Manual**.)
- To see a walk-through example of particles energy spectra being using to analyze breakage in a SAG mill, review Tutorial - SAG Mill in the *Rocky Tutorial Guide*.

Viewing Particles Energy Spectra

If prior to processing your simulation, you enabled **Particles Energy Spectra** to be collected, then while the simulation is processing, Rocky calculates the selected Cumulative Specific Power curves (Dissipation, Impact, and/or Shear) for each particle set (p. 1393), generated by each size of the particle size distribution, and separated by Specific Energy. These curves are grouped under Specific Energy on the Curves tab (Figure 6).

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

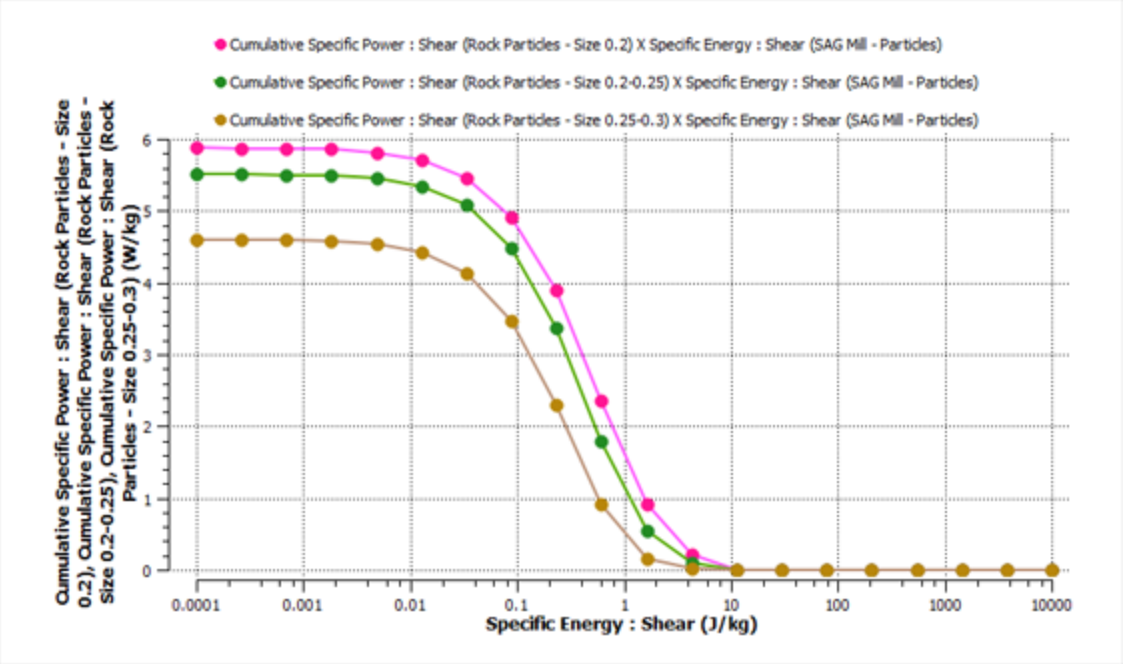
Figure 2.217: Curves tab for Particles showing three sections for Specific Energy



Using these curves, you can then create a Cross plot, similar to the example below (Figure 7).

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

Figure 2.218: Example cross plot showing Specific Shear Energy Curves



Tip:

For more information about these Curves, see About Curves (p. 989).

About Opening Older Energy Spectra Projects

When opening projects that were created and processed in versions of Rocky prior to 2022 R2, Rocky will still read the energy spectra data in those projects and will display the energy spectra Curves as usual.

However, if you want to modify and then re-process an older energy spectra simulation in 2022 R2 or later versions, you will need to enable the energy spectra modules and define the type of collection parameters you want prior to processing the simulation.

The same is true for projects that you have created but not processed in older versions of Rocky; you must enable and define the energy spectra modules manually before processing your simulation.

Particles Energy Spectra Collection Options

Use Figures 1-2 above and the tables below to understand how to collect particle-based energy statistics.

Table 1: Modules, Particles Energy Spectra parameter definitions

Setting	Description	Range
Dissipation Energy	When enabled, collects particle-based statistics for dissipation energy, which is the fraction of the mechanical energy transformed irreversibly into other forms of energy during a collision.	Turns on or off

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

Setting	Description	Range
Impact Energy	When enabled, collects particle-based statistics for impact energy, which is the maximum energy transferred during a collision. **Tip:** When Rocky simulates an actual breakage process (see also Enable and View Particle Breakage), this is usually the energy type considered to quantify the damage on the particles.	Turns on or off
Shear Energy	When enabled, collects particle-based statistics for shear energy, which is the work done by the tangential contact forces during a collision.	Turns on or off
Number of Bins	This defines how many sub-intervals the specified energy interval is evenly (using a logarithmic scale) subdivided. Determines the the resolution of the resulting Curves. The higher the value, the better the resolution.	Positive values
Maximum Specific Energy	This defines the right bound of the interval of specific energies that will be subdivided into bins for the construction of the energy spectra Curves.	Positive values
Minimum Specific Energy	This defines the left bound of the interval of specific energies that will be subdivided into bins for the construction of the energy spectra Curves.	Positive values
Start Time	The amount of time you want to wait before starting to register energy spectra data. **Tip:** It is best practice to set your **Start Time** to begin after a steady state has been reached in your particle flow.	Value must be positive but less than **Simulation Duration**, which is located on the **Solver | Time** sub-tab. (See also About Solver Parameters.)
Time Delay after Release	The amount of time you want to wait after a particle has been released before starting to register energy spectra data.	Value must be positive but less than **Simulation Duration**, which is located on the **Solver | Time** sub-tab. (See also About Solver Parameters.)

Table 2: Individual Particle Group, Modules sub-tab parameter definitions

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

Setting	Description	Range
Enable For This Particle Group	When enabled, the energy values involving particles from this group will be recorded for energy spectra purposes. When cleared, the energy values involving particles from this group will be ignored.	Turns on or off

What would you like to do?

- Enable and View Data for Particles Energy Spectra (p. 369)
- Learn more About Collecting Data (p. 306)
- Learn more About Curves (p. 989)

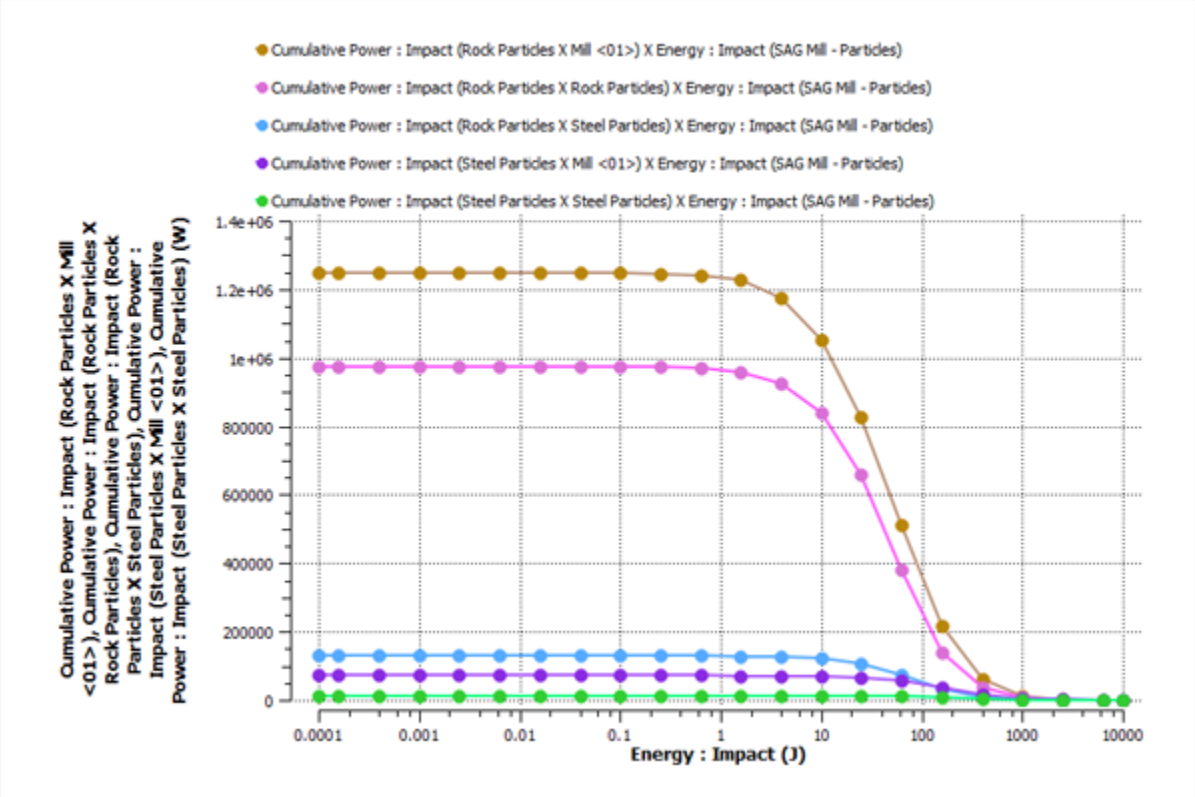
See Also

- About Contacts Energy Spectra (p. 350)
- View Frequently Used Tasks (p. 371)

ENABLE AND VIEW DATA FOR CONTACTS ENERGY SPECTRA

Analyzing data for Contacts Energy Spectra involves first turning on the type of collection you want prior to processing the simulation (see also About Contacts Energy Spectra (p. 350)), and then determining which particle groups and geometries you want to involve in that collection. After processing your simulation you can then plot the resulting Curves (Figure 1).

Figure 2.219: Example cross plot showing energy spectra values



It is important to realize that data is only collected after the **Start Time** value is reached.

For more details, tips, and limitations, see the About Contacts Energy Spectra (p. 350) topic.

To Enable and View Data for Contacts Energy Spectra:

1. Set up your simulation as you normally would (see also Set Simulation Parameters (p. 414)).
2. Before processing your simulation, do all of the following:
    a. From the **Data** panel, select **Modules**, and then from the **Data Editors** panel, select **Contacts Energy Spectra**.
    b. From the **Data** panel, under **Modules**, select the new **Contacts Energy Spectra** entry.
    c. From the **Data Editors** panel, on the **Contacts Energy Spectra** tab, do all of the following:
        i. Select one or more of the checkboxes for the type of energy statistics you want collected. (See also About Contacts Energy Spectra (p. 350).) Important: You must select at least one checkbox in order for energy spectra to be collected.
        ii. In the **Number of Bins** box, enter the number of sub-intervals you want the energy interval to be evenly (using a logarithmic scale) subdivided.
        iii. In the **Maximum Energy** and **Minimum Energy** boxes, enter values for the right an left bounds of the interval of energies that will be subdivided into bins for the construction of the energy spectra Curves.
        iv. Set the values you want for both **Start Time** and **Time Delay after Release**. IMPORTANT: Because energy spectra is calculated as an average over time, it is important that you set this data to be collected after steady state has been reached.
    d. For each geometry component that you do *not* want included in the collection of energy spectra, do the following:
        i. From the **Data** panel, under **Geometries** select the geometry that you want to exclude from energy spectra collections.
        ii. From the **Data Editors** panel, select the **Geometry | Modules** tab and then under **Contacts Energy Spectra** clear the **Enable For This Geometry** checkbox.
    e. For each particle group that you do *not* want included in the collection of energy spectra, do the following:
        i. From the **Data** panel, under **Particles** select the particle group that you want to exclude from energy spectra collections.
        ii. From the **Data Editors** panel, select the **Particle | Modules** tab and then under **Contacts Energy Spectra** clear the **Enable For This Particle Group** checkbox.
3. Process your simulation as you normally would. (See also About Starting a Simulation (p. 908).)
4. After your simulation reaches as least as far as the **Start Time** value, do the following:
    a. From the **Data** panel, select **Particles**.

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

b. From the **Data Editors** panel, select the **Curves** tab.

c. Under the three **Energy** categories (Dissipation, Impact, and Shear), drag one or more of the Curves listed to the Workspace. (See also About Curves (p. 989).) A new Cross Plot window is created showing the Curve(s) you selected.

d. From the Cross Plot window, do one or both of the following:

- Modify the appearance of the Cross Plot. (See also About Changing the Appearance of a Graph (Plot or Histogram) (p. 1126).) Tips: To show a Curve in a logarithmic format, right-click the desired axis and select **Customize Axis**. From the **Axis** tab, under **Scale Options**, enable the **Logarithmic Scale** checkbox.
- Export the data to a CSV file. (See also Export Data into a CSV File (p. 1164).)

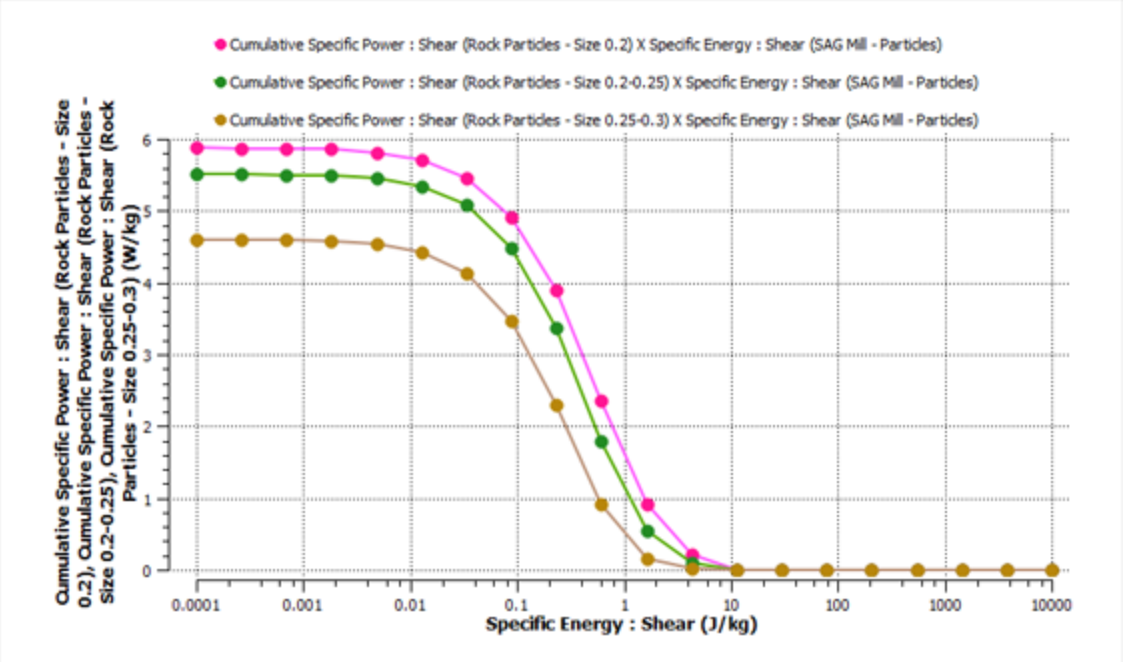
See Also:

- About Contacts Energy Spectra (p. 350)
- About Collecting Data (p. 306)
- About Curves (p. 989)
- About Solver Parameters (p. 464)

ENABLE AND VIEW DATA FOR PARTICLES ENERGY SPECTRA

Analyzing data for Particles Energy Spectra involves first turning on the type of collection you want prior to processing (see also About Particles Energy Spectra (p. 358)), and then determining which particle groups you want to involve in that collection. After processing your simulation you can then plot the resulting Curves (Figure 1).

Figure 2.220: Example cross plot showing energy spectra values



It is important to realize that data is only collected after the **Start Time** value is reached.

For more details, tips, and limitations, see the About Particles Energy Spectra (p. 358) topic.

To Enable and View Data for Particles Energy Spectra:

1. Set up your simulation as you normally would (see also Set Simulation Parameters (p. 414)).
2. Before processing your simulation, do all of the following:
    a. From the **Data** panel, select **Modules**, and then from the **Data Editors** panel, select **Particles Energy Spectra**.
    b. From the **Data** panel, under **Modules**, select the new **Particles Energy Spectra** entry.
    c. From the **Data Editors** panel, on the **Particles Energy Spectra** tab, do all of the following:
        i. Select one or more of the checkboxes for the type of energy statistics you want collected. (See also About Particles Energy Spectra (p. 358).) Important: You must select at least one checkbox in order for energy spectra to be collected.
        ii. In the **Number of Bins** box, enter the number of sub-intervals you want the specific energy interval to be evenly (using a logarithmic scale) subdivided.
        iii. In the **Maximum Energy** and **Minimum Energy** boxes, enter values for the right an left bounds of the interval of specific energies that will be subdivided into bins for the construction of the energy spectra Curves.
        iv. Set the values you want for both **Start Time** and **Time Delay after Release**. IMPORTANT: Because energy spectra is calculated as an average over time, it is important that you set this data to be collected after steady state has been reached.
    d. For each particle group that you do *not* want included in the collection of energy spectra, do the following:
        i. From the **Data** panel, under **Particles** select the particle group that you want to exclude from energy spectra collections.
        ii. From the **Data Editors** panel, select the **Particle | Modules** tab and then under **Particles Energy Spectra** clear the **Enable For This Particle Group** checkbox.
3. Process your simulation as you normally would. (See also About Starting a Simulation (p. 908).)
4. After your simulation reaches as least as far as the **Start Time** value, do the following:
    a. From the **Data** panel, select **Particles**.
    b. From the **Data Editors** panel, select the **Curves** tab.
    c. Under the three **Specific Energy** categories (Dissipation, Impact, and Shear), drag one or more of the Curves listed to the Workspace. (See also About Curves (p. 989).) A new Cross Plot window is created showing the Curve(s) you selected.
    d. From the Cross Plot window, do one or both of the following:

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

- Modify the appearance of the Cross Plot. (See also About Changing the Appearance of a Graph (Plot or Histogram) (p. 1126).) Tip: To show a Curve in a logarithmic format, right-click the desired axis and select **Customize Axis**. From the **Axis** tab, under **Scale Options**, enable the **Logarithmic Scale** checkbox.
- Export the data to a CSV file. (See also Export Data into a CSV File (p. 1164).)

See Also:

- About Particles Energy Spectra (p. 358)
- About Collecting Data (p. 306)
- About Curves (p. 989)

VIEW FREQUENTLY USED TASKS

Use this section to quickly find and perform common tasks in Rocky.

What would you like to do?

- Enter Input Variables or Mathematical Functions as Parameter Values (p. 372)
- Change Set Up Parameters in a Partially Processed Simulation (p. 846)
- Change the Speed of a Default Conveyor (p. 373)
- Change Particle Adhesion (p. 374)
- Set Particle Size Ranges (p. 375)
- See Surface Wear on the Geometry Itself (p. 378)
- Import a Custom Particle Shape (p. 670)
- Use the 1-Way Fluent Method to Process Fluent and Rocky Simulations (p. 383)
- Use the 2-Way Fluent Method to Process Fluent and Rocky Simulations (p. 386)
- Enable Particle Breakage Calculations (p. 397)
- Create and View a Particle Assembly (p. 398)
- Enable Thermal Modeling Calculations (p. 400)
- Add a Logo to a 3D View (p. 402)
- Reuse 3D View Window, Workspace, and User Process Setups in Other Projects (p. 403)
- Undo or Redo Several Actions at Once (p. 108)
- Set Up a New GPU Card (p. 404)

See Also:

- Set Your Rocky Preferences (p. 226)

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

ENTER INPUT VARIABLES OR MATHEMATICAL FUNCTIONS AS PARAMETER VALUES

You can create Input variables in Rocky (see Defining Your Variables (p. 243)) and then use them in place of actual numerical values in situations where you want more flexibility. These variables can be used alone or within functions.

Tip:

Rocky includes several visual feedback mechanisms, like text color, to help you ensure the variable names and mathematical functions you enter are correct. See Use Colored Text to Validate the Syntax of Your Entries (p. 117) for more information.

Important:Verify that your field units are set the way you want before using a variable-or function using a variable-within a field. Even though variables are always saved in the Variables list without units, as soon a variable-or function using a variable-is used within a parameter field, it takes on whatever units are currently set for that field and then those units are recorded for that specific usage in the Expressions list. Changing the units for the field after the variable or function have already been entered only converts the numerical display to reflect the new units; the original variable or function amount and units are still retained. However, you can change the units and value later by editing it in the Expressions list.

Most text fields in Rocky support entering variables or mathematical functions as parameter variables, including three-part text fields. There are a few exceptions to be aware of, however. (See also I cannot enter an input variable or mathematical function into a text field (p. 1334).)

Most of these parametric expressions will be retained while saving a copy of a project for restart purposes, but there are a few exceptions for some **Start Time** and **Stop Time** fields. (See also I get a "Links removed" message when I save my project for restart purposes (p. 1381).)

Most text fields will also support the entering of pre-defined mathematical functions (e.g., *floor* and *pow*) and constants (e.g., *pi* and *e*) that come standard with Python-based programs, including Rocky. And as such, these particular functions and constants cannot be used when defining Input variables.

What would you like to do?

- Replace a Parameter Value with a Variable (p. 372)
- Replace a Parameter Value with a Function Using a Variable (p. 373)
- Replace a Parameter Value with a Pre-Defined Mathematical Constant (p. 373)

See Also:

- View Frequently Used Tasks (p. 371)
- I cannot enter an input variable or mathematical function into a text field (p. 1334)

REPLACE A PARAMETER VALUE WITH A VARIABLE

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

- Using the **Data** and **Data Editors** panels, locate and select the field for the parameter you want to replace, exactly enter the name of the variable you want to use, and then press Enter. The current value set for the variable in the **Expressions/Variables** panel is displayed in the field.

Tip:

To see the units that are set for the variable you used, double click the value in the field. The units are displayed in brackets [ ] after the variable name.

Note:

You cannot use pre-defined functions or constants to define your variable.

See Also:

- Enter Input Variables or Mathematical Functions as Parameter Values (p. 372)
- Replace a Parameter Value with a Function Using a Variable (p. 373)
- Replace a Parameter Value with a Pre-Defined Mathematical Constant (p. 373)

REPLACE A PARAMETER VALUE WITH A FUNCTION USING A VARIABLE

- Using the **Data** and **Data Editors** panels, locate and select the field for the parameter you want to replace, and then enter the function you want to use.

See Also:

- Enter Input Variables or Mathematical Functions as Parameter Values (p. 372)
- Replace a Parameter Value with a Variable (p. 372)
- Replace a Parameter Value with a Pre-Defined Mathematical Constant (p. 373)

REPLACE A PARAMETER VALUE WITH A PRE-DEFINED MATHEMATICAL CONSTANT

- Using the **Data** and **Data Editors** panels, locate and select the field for the parameter you want to replace, and then enter the pre-defined mathematical function or constant you want to use.

See Also:

- Enter Input Variables or Mathematical Functions as Parameter Values (p. 372)
- Replace a Parameter Value with a Variable (p. 372)
- Replace a Parameter Value with a Function Using a Variable (p. 373)

CHANGE THE SPEED OF A DEFAULT CONVEYOR

You can change the speed of a default receiving or feed conveyor at any point during the simulation. For example, you might want to measure reclaiming belt power by starting off a simulation with a feed conveyor stopped until a hopper is filled with particles, and then ending the simulation with the conveyor up to full speed.

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

TO CHANGE THE SPEED OF A DEFAULT CONVEYOR:

1. Ensure the default conveyor for which you want to change belt speed has already been added. (See also Add and Edit Geometry Components (p. 480).)

2. From the **Data** panel, under **Geometries**, select the **Receiving Conveyor** or **Feed Conveyor** for which you want to change belt speed. The parameters for that geometry are displayed in the **Data Editors** panel.

3. From the **Data Editors** panel, on the **Receiving Conveyor** or **Feed Conveyor** tab, ensure the **Belt Motion** sub-tab is selected, and then do all of the following:

    a. In the **Belt Speed** box, enter the full speed you want the belt to reach after acceleration is complete.

    b. In the **Beginning Start Time** box, enter the amount of time that you want the simulation to run before the belt starts accelerating.

    c. In the **Acceleration Period** box, enter the amount of time you want the belt to increase from full stop to full **Belt Speed**.

    d. In the **Beginning Stop Time** box, enter the amount of time that you want the simulation to run before the belt starts decelerating.

    e. In the **Deceleration Period** box, enter the amount of time that you want the belt to decrease from full **Belt Speed** to full stop.

See Also:

- View Frequently Used Tasks (p. 371)

CHANGE PARTICLE ADHESION

You change particle adhesion when you want a particle to act wetter or "stickier" than normal during a simulation. This might be useful in showing how ore flows through handling equipment during the wet season as compared to the dry, for example.

Tip: It is good practice to calibrate your particle settings to the real-world behavior of the material you are simulating before changing settings like adhesion. You may wish to use the Material Wizard and/or the Calibration Suite to help you with this task.

TO CHANGE PARTICLE ADHESION:

1. Set up the simulation as you normally would. (See also Setting Up a Simulation (p. 411).)

2. During the Set Simulation-Wide Parameters step, ensure you select both the **Rolling Resistance Model** and the **Adhesive Force** you want from the **Physics** | **Momentum** tab. (See also About Physics Parameters (p. 417).)

3. Adjust Materials Interactions values by doing all of following:

    a. From the **Data** panel, select **Materials Interactions**.

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

b. From the **Data Editors** panel, choose the materials you want to modify from the **Select Materials** lists.

c. Adjust one or all of the settings provided including friction, adhesive distance, and force and/or stiffness fractions. (See also About Modifying Materials Interactions and Adhesion Values (p. 597).)

4. Adjust Rolling Resistance values by doing all of the following:

a. From the **Data** panel, under **Particles**, select the rigid, convex Particle set (p. 1393) for which you want to change adhesion.

b. From the **Data Editors** panel, select the **Movement** tab, and then adjust the **Rolling Resistance** value. **Notes**: - The **Type C: Linear Spring Rolling Limit** Rolling Resistance model is recommended for sticky particles. (See also About Physics Parameters (p. 417).) - Rolling Resistance is not usually needed for shaped particles (p. 1395).

5. Process the simulation and then continue to adjust the Materials Interactions and Rolling Resistance values until you get the results you want.

See Also:

- View Frequently Used Tasks (p. 371)

SET PARTICLE SIZE RANGES

You set a particle size range-otherwise known as a Particle Size Distribution (PSD)-when you want particles of one shape but different sizes to be used in the simulation. If you set more than one size for a Particle set, Rocky uses a calculation to specify the distribution of sizes within any two dimensions you specify. This helps to more accurately reflect real matter.

When setting size ranges, the following three methods, or **Size Types**, can be used as a basis for defining your particle sizes (Figure 1):

- **Sieve Size** (formerly "Sieve"), which for especially default Rocky shapes, bases particle size upon a virtual mesh hole just big enough for the particle to pass through. **Note**: Due to the way Sieve Size is calculated, it should be used only for even and balanced particle shapes (such as the default **Solid** particles included with Rocky) and not for fibers or shapes with high aspect ratios. (For more detail on how Sieve Size is calculated in Rocky, see the DEM Technical Manual. (From the Rocky **Help** menu, point to **Manuals**, and then click **DEM Technical Manual**.))

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

- **Equivalent Sphere Diameter** (formerly "Equivalent Diameter"), which for especially irregular objects, bases particle size upon the diameter of a sphere with equivalent volume.

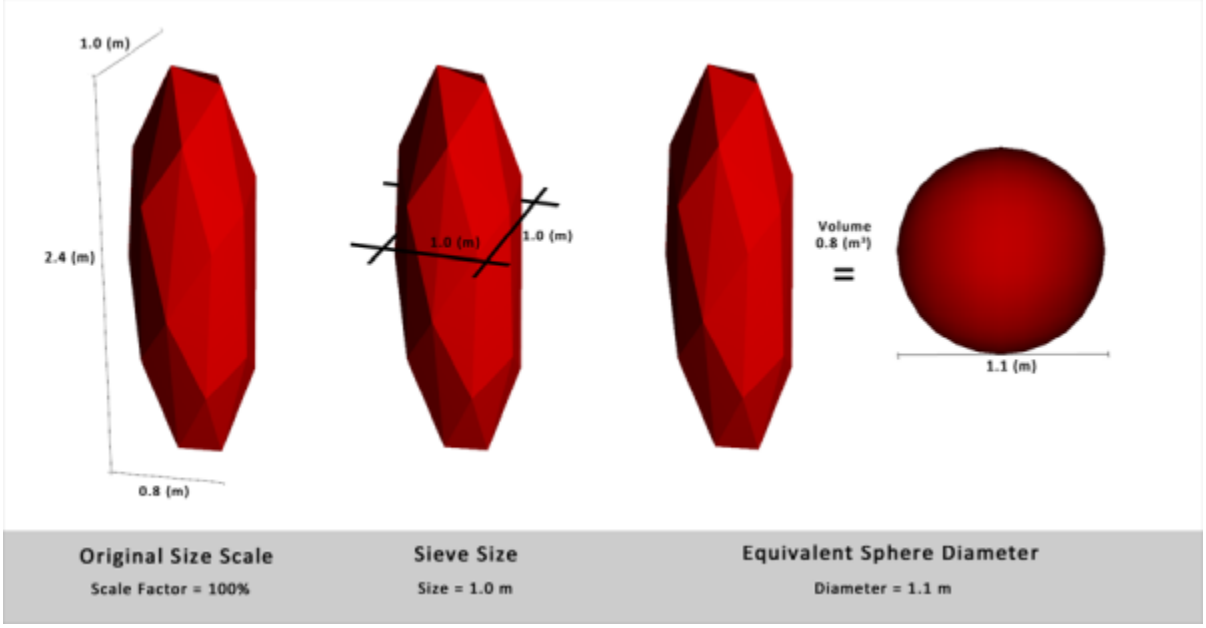
Tip:

To see a walk-through example of setting and using Equivalent Sphere Diameter, refer to the following Tutorial: Tutorial - Windshifter in the *Rocky Tutorial Guide*. **Original Size Scale**, which for especially imported shapes, bases particle size upon the original size of the shape.

Tip:

To see a walk-through example of setting and using Original Size Scale, refer to the following Tutorial: Tutorial - Tablet Coater in the *Rocky Tutorial Guide*.

Figure 2.221: Comparison of the three different Size Types in Rocky



The PSD in Rocky is linear by Particle Mass or, on a semi-log scale, by Particle size and not by number. When you enter the percentage of cumulative mass and then either particle sieve **Size**, sphere **Diameter**, or original size **Scale Factor** (representing the three available **Size Types**, respectively), these represent points on a semi-log plot. Anything between two points follows the linear rule:

$$Mass = A + B * \log(\text{Size})$$
$$Mass = A + B * \log(\text{Diameter})$$
$$Mass = A + B * \log(\text{Scale Factor})$$

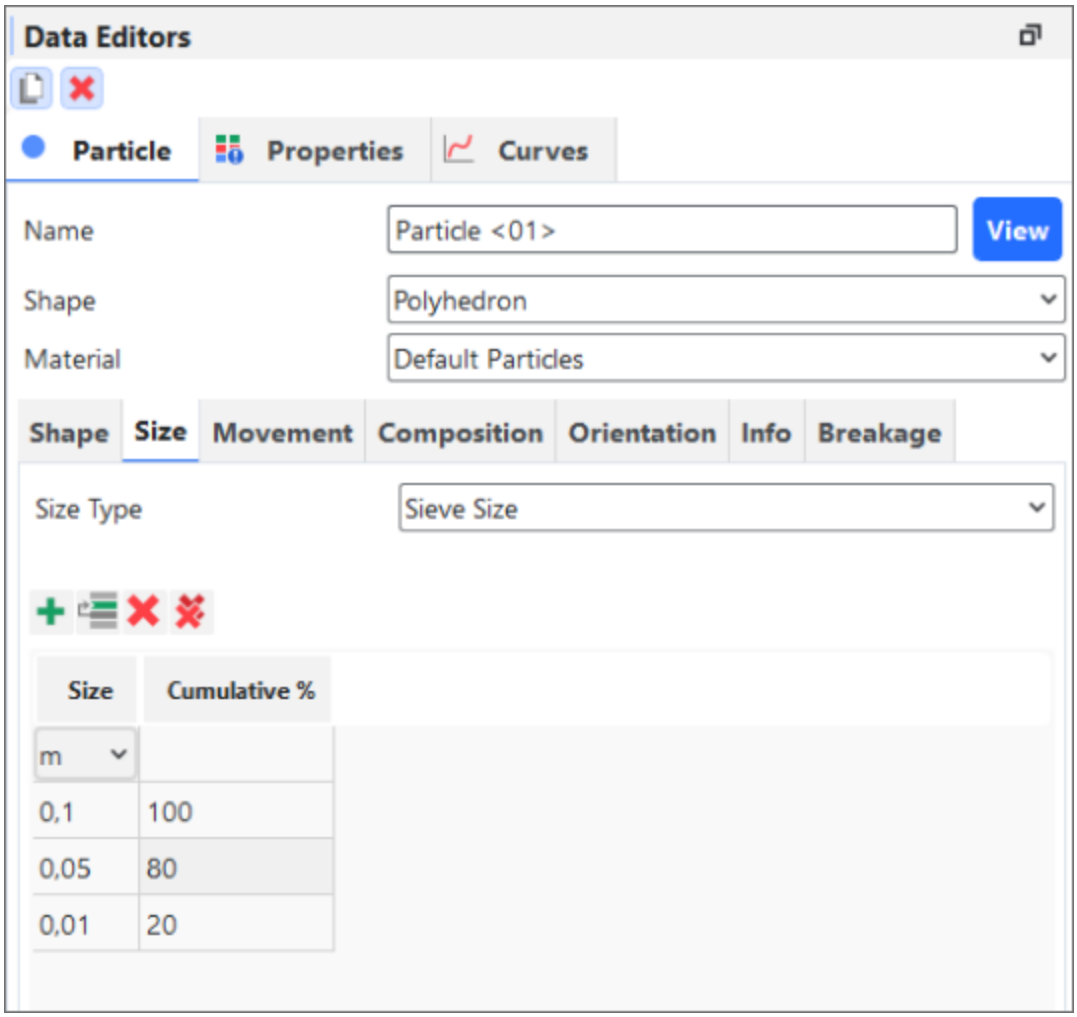
(See also About Adding and Editing Particle Sets (p. 602).)

For example, if you specify 0.1 and 0.05 (m) particle sizes measured by sieve size, you will get a range of various particle sizes within those two dimensions.

Use the image and procedure below to set size ranges for your Particle sets.

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

Figure 2.222: Example Size Distribution Settings for Sieve Size



TO SET A PARTICLE SIZE RANGE OR PSD:

1. Ensure that the Particle set for which you want to specify size ranges has been added. (See also Add a New Particle Set (p. 678).)

2. From the **Data** panel, select the Particle set you want (for example, Particle <01>) and then from the **Data Editors** panel, select the **Size** tab.

3. Set what you want for **Size Type**. (See also About Adding and Editing Particle Sets (p. 602).)

4. Click the **Add** button once for each additional size range you require. **Note**: For rigid particles composed of Single Elements, there is no limit to the amount of rows you can add. For flexible or rigid particles composed of Multiple Elements (also known as *meshed* (p. 1392) particles-see the **Composition Tab** section in the About Adding and Editing Particle Sets (p. 602) topic), only one row can be defined per Particle set.

5. Under **Size**, **Diameter**, or **Scale Factor** respectively, and starting with the first row, enter the maximum and minimum size values for each range in decreasing order. *For example*, to get a range of sieve sizes between 0.05-0.1 m and 0.01-0.05 m, enter 0.1, 0.05, and 0.01 in the first, second, and third rows respectively. (See Figure 2 above for example.) **Note**: The smallest value you enter (last row) will also be shown at that exact size (no range).

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

6. Under **Cumulative %**, starting with the second row, indicate what percentage of the total particle mass you want of that size value range by entering in decreasing order the previous row's percentage minus the percentage you want. *For example*, if you want 20% of the 0.05-0.1 m range, enter 80 (100-20); to get 60% of the 0.01-0.05 m range, enter 20 (80-20). (See Figure 2 above for example.)

Note: The percentage indicated in the final row also indicates what particle mass amount will be of that exact size (no range). For example, if the smallest size you enter is 0.01 m and you have entered 20 for **Cumulative %**, 20% of the particle mass will be exactly 0.01 m in size in addition to the ranges you have set.

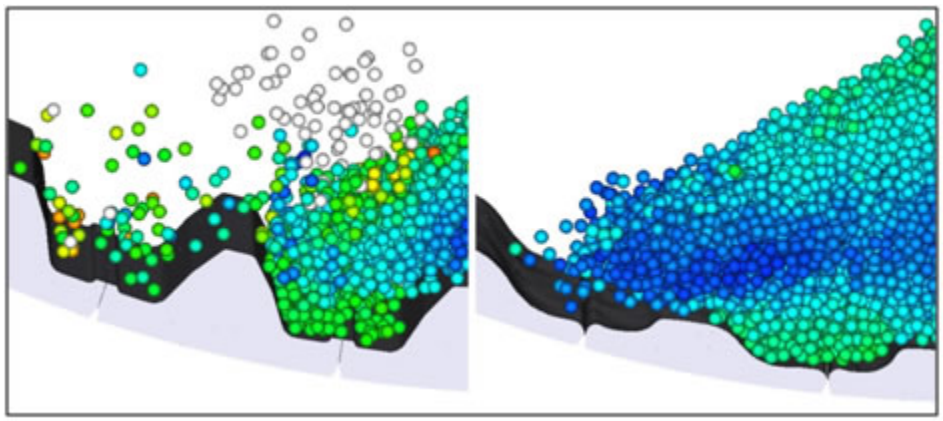
See Also:

- View Frequently Used Tasks (p. 371)

SEE SURFACE WEAR ON THE GEOMETRY ITSELF

There are two primary ways you can use Rocky to gain an understanding of how your geometries will wear over time. One way is by enabling surface wear modification, which changes the physical appearance of the geometry as the simulation progresses (Figure 1).

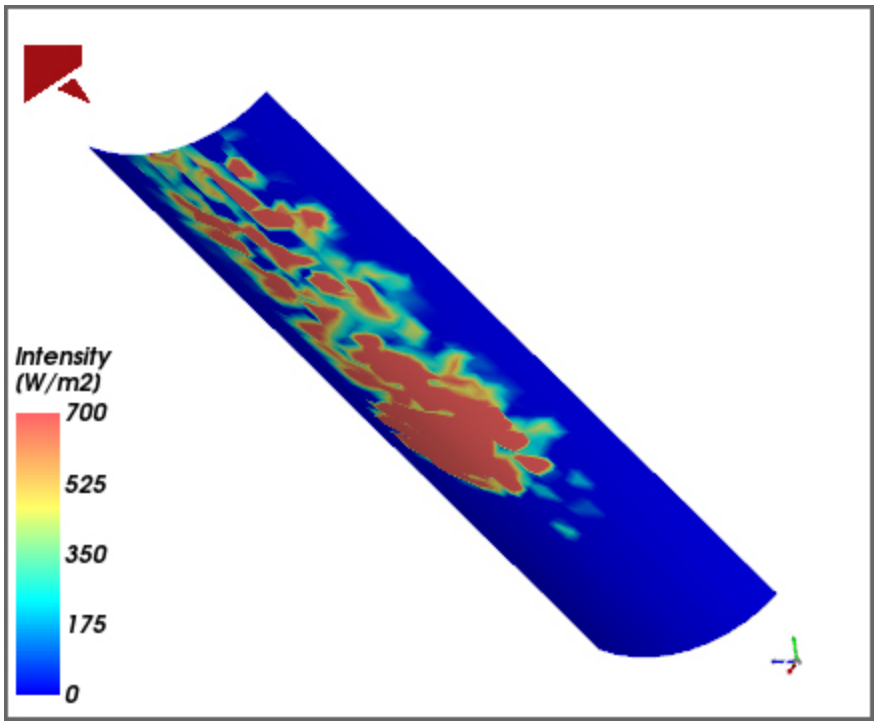
Figure 2.223: Surface wear modification



A second way is to turn on Boundary Collision Statistics (see also About Collision Statistics for Boundaries (p. 312) and then view a color map of the surface intensity (Figure 2), impact velocity, stresses, and more.

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

Figure 2.224: Color map of surface intensity



These features are independent of each other and are enabled from different areas of the Rocky UI. However, to view either type of wear data, you need to ensure that you have enabled enough geometry triangles prior to processing your simulation. Too few triangles will result in a jagged or blocky representation rather than a smooth one, but too many triangles can slow down your simulation.

Tip: To see a walk-through example of surface wear modification being using to analyze a SAG mill, review Tutorial - SAG Mill in the *Rocky Tutorial Guide*.

What would you like to do?

- View Surface Wear Modification on an Imported Geometry (p. 379)
- View a Color Map of Wear on the Default Belt or Imported Geometry Itself (p. 381)
- Learn more About Collision Statistics for Boundaries (p. 312)

See Also:

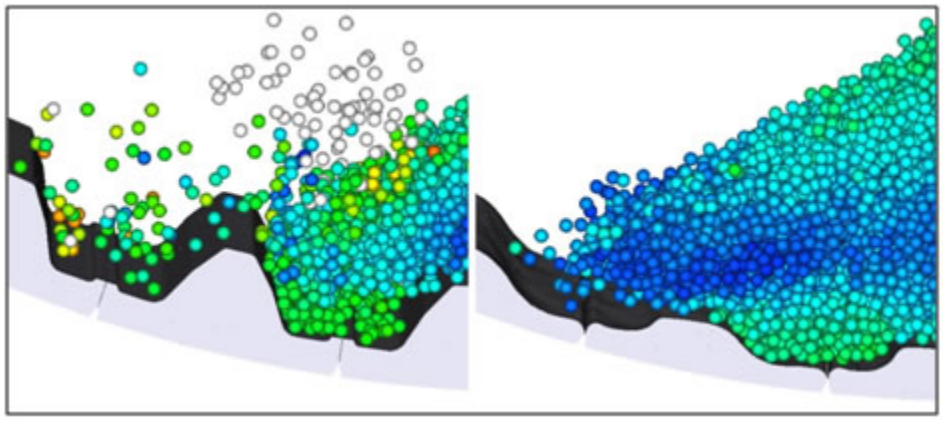
- View Frequently Used Tasks (p. 371)
- Understand Data Collection in Rocky (p. 306)

ENABLE AND VIEW SURFACE WEAR MODIFICATION ON AN IMPORTED GEOMETRY

One way you can use Rocky to gain an understanding of how your geometries will wear over time is by enabling surface wear modification. Turning this feature on for an imported geometry prior to processing changes the physical appearance of the geometry as the simulation progresses (Figure 1).

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

Figure 2.225: Surface wear modification



To view this type of wear data, you must ensure that you have enabled enough geometry triangles prior to processing your simulation. Too few triangles will result in a jagged or blocky representation rather than a smooth one, but too many triangles can slow down your simulation.

It is important to realize that this data is only collected after the Wear **Start** value on the **Solver** | **Time** tab is reached. This is to allow enough time for the particle flow within your simulation to reach a steady state before surface wear modification calculations begin.

Tips: To see walk-through examples of surface wear modification, refer to the following Tutorials:

- Tutorial - SAG Mill in the *Rocky Tutorial Guide*
- Tutorial - High Pressure Grinding Roll (HPGR) in the *Rocky Tutorial Guide*

TO ENABLE AND VIEW SURFACE WEAR MODIFICATION:

1. Set up the simulation as you normally would. (See also Set Simulation Parameters (p. 414)). **Tip**: For easier comparison after the simulation is complete, consider including a copy of the imported geometry that will remain unworn. (See also Duplicate a Data Panel Item (p. 846).)

2. Before processing the simulation, do all of the following:

    a. From the **Data** panel, under **Geometries**, select the imported geometry for which you want to enable surface wear. The parameters for the geometry become active in the **Data Editors** panel.

    b. From the **Data Editors** panel, on the **Geometry** tab, do all of the following:

        i. From the **Geometry** sub-tab, ensure **Triangle Size** is small enough to enable the wear detail you want. (0.1 m is recommended for most chutes and mills).

        ii. From the **Wear** sub-tab, select the **Wear Model** you want to use, and then enter the **Volume/Shear Work Ratio**. Note: This is a calibration step that comes from real-world wear data you have collected on similar types of equipment.

    c. From the **Solver** | **Time** tab, ensure that your Wear **Start** time is set to when you want wear data to be collected. (From the **Data** panel, select **Solver** and then from the **Data Editors** panel, locate this value on the **Time** sub-tab.) (See also About Solver Parameters (p. 464).) **Tip**: It is best practice to set your Wear **Start** time to begin after a steady state has been reached for your particle flow.

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

3. Process the simulation as you normally would. (See also Processing a Simulation (p. 907).)

4. From a 3D View window (see also About 3D View Windows (p. 1040)), observe the changes to the surfaces of the geometry as the simulation progresses. **Tip**: If you included a copy of the original geometry in the simulation, you can use the unworn geometry as a comparison to one worn by surface wear modification.

Tips:

- If you want to analyze the modified geometry outside of Rocky, you can choose to export it to an STL file. (See also Export a Geometry Component to an STL File (p. 520).)

- It is possible to save a copy of an in-process simulation with surface wear and have the wear calculations continue in the copy. (See also Save a Copy of a Partially Processed Simulation for Restart Purposes (p. 855).) While you are unable to turn off wear calculations from the copy, if you no longer need wear, you can get around this by setting **Volume/Shear Work Ratio** to zero.

- It is also possible to have wear calculations be made on replicated geometries.

See Also:

- About Collecting Data in Rocky (p. 306)

- Enable and View Collision Statistics for Boundaries (p. 318)

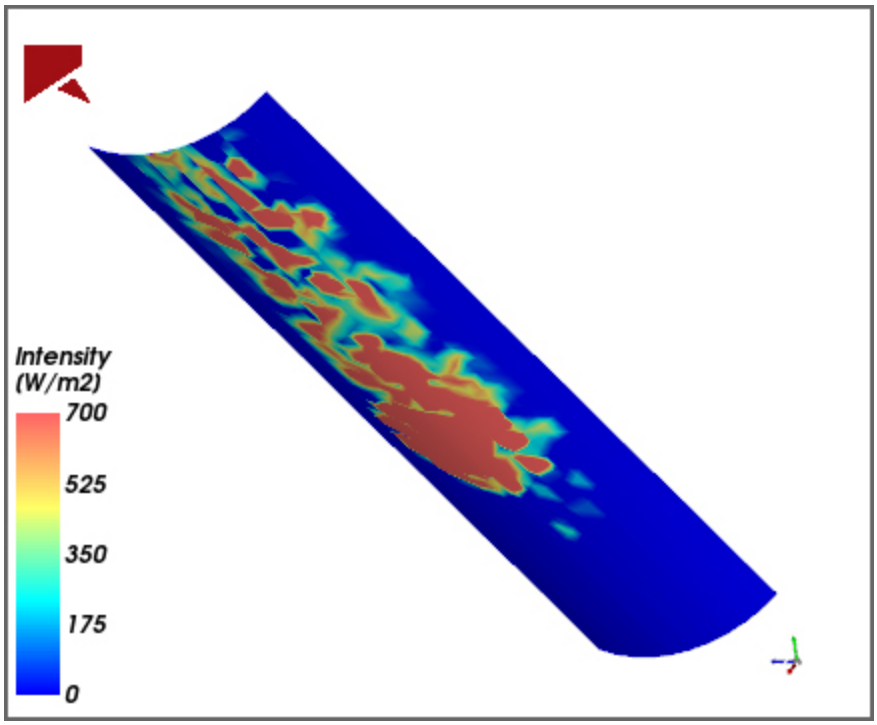
- Export a Geometry Component to an STL File (p. 520)

VIEW A COLOR MAP OF WEAR ON THE DEFAULT BELT OR IMPORTED GEOMETRY ITSELF

When you choose to turn on Boundary Collision Statistics (see also About Collision Statistics for Boundaries (p. 312) and enable wear-related categories such as **Intensities**, you are able to create and view a color map of the surface intensity (Figure 1), impact velocity, stresses, and more. These color maps can help you analyze the causes of excess wear on your conveyor belts or imported geometries.

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

Figure 2.226: Color map of surface intensity



TO VIEW A COLOR MAP OF WEAR ON THE DEFAULT BELT OR IMPORTED GEOMETRY ITSELF

1. Set up the simulation as you normally would. (See also Set Simulation Parameters (p. 414).)

2. Before processing your simulation, ensure all of the following:

   a. You have enabled the Boundary Collision Statistics that you want to be collected. (See also Enable and View Collision Statistics for Boundaries (p. 318).)

   b. You have verified that the default conveyor or imported geometry has its **Triangle Size** set small enough to enable the wear detail you want. (0.1 m is recommended for most chutes and mills). (From the **Data** panel under **Geometry**, select the component you want to verify. From the **Data Editors** panel, on the **Geometry** sub-tab, verify the **Triangle Size** value.)

3. Process the simulation as you normally would. (See also Processing a Simulation (p. 907).)

4. Ensure you have a 3D View displayed in the Workspace. (See also Create and Modify a 3D View (p. 1040).)

5. When processing is complete enough to show the wear you want to visualize, from the first drop down list on the **Coloring Settings** toolbar, choose the data you want to display (e.g., **Intensity**.) The geometries in your simulation change color and a color legend is displayed in the 3D View.

6. Do any or all the following to adjust the view and color map details:

   - To better focus on the area you want to view, use the mouse to pan, rotate, and zoom. (See also About Using the Mouse and Keyboard to Change the View (p. 1043).)

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

- To hide all items but the belt or geometry you want to focus on, use the eye icons to the left of the geometries and particles in the **Data** panel. (See also Show/Hide Components by Using Eye icons and Checkboxes (p. 113).)
- To modify the color bar, right-click the color legend and change the options listed. (See also About Color Scales (p. 1086).)
- To focus on a different time period within the simulation, use the arrows or slider on the Time toolbar. (See also About the Time Toolbar (p. 1279).)

See Also:

- Enable and View Collision Statistics for Boundaries (p. 318)
- Enable and View Surface Wear Modification on an Imported Geometry (p. 379)
- About Collision Statistics for Boundaries (p. 312)
- Understand Data Collection in Rocky (p. 306)

USE THE 1-WAY FLUENT METHOD TO PROCESS FLUENT AND ROCKY SIMULATIONS

There are five main steps to using the 1-Way Fluent CFD Coupling method to process a 1-Way coupled Rocky and Ansys Fluent simulation, as described below.

Tips:

- You may also complete this process from within Ansys Workbench. To accomplish this with transient Fluent data, however, there are separate steps you must follow. (See also Set Up and Run a 1-Way Fluent Transient Project within Ansys Workbench (p. 789).)
- You may also complete this process without using the Fluent UI. (See also Use a Journal to Run Fluent and Export Transient 1-Way Results (p. 1313).)

STEP I: VERIFY THAT ALL ROCKY-FLUENT PROGRAM REQUIREMENTS ARE MET

1. Before beginning Step II below, ensure all of the following program requirements have been met:
   - A Rocky-supported version of Ansys Fluent is installed upon a machine that also contains a Rocky program installation. (See also System Requirements (p. 67).) Note: Rocky does not need Fluent to be installed on the same machine in order to run a 1-Way Fluent simulation. However, Fluent needs Rocky to be installed on the same machine in order for Rocky to enable the **Rocky Export** menu item in Fluent.
   - If you are not certain whether you have installed the **Ansys Fluent Coupling Support** component during Rocky installation, first follow the Install Ansys Coupling Components (p. 792) procedure before beginning Step II.

STEP II: SET UP, PROCESS, AND THEN EXPORT THE CFD SIMULATION

1. In Ansys Fluent, set up the CFD simulation that you want to couple with Rocky. Important: Ensure that you complete all of the following; these exact settings are required for 1-Way Fluent coupled simulations:

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

a. From the **Fluent Launcher** dialog, do the following:

   i. Under **Dimension** ensure **3D** is selected.

   ii. Under **Options** select **Double Precision**.

b. From the **Solution Methods** tab, under **Pressure-Velocity Coupling Scheme**, ensure **SIMPLE** is selected.

Optional Steps:

- If you want to use any standard Fluent post processing options, set them up now before processing the simulation.
- If you want both sides of your coupled simulation to share identical geometries, set up those geometries now in Fluent, but be sure you save your Fluent Case File. Later in Rocky, you can choose to import those geometries directly from the CAS, CAS.H5, or CAS.GZ file that you will save from Fluent in the next step.
- If you want both sides of your coupled simulation to share the same geometry movements, set up the translation or rotation movements now using Fluent Moving Meshes. These movements will later be imported (with the F2R file) into Rocky (Step IV.2 below), which will automatically convert them into Rocky Motion Frames. Note: If you chose to define your moving meshes in Fluent using expressions, you will need to set up your Motion Frames in Rocky manually. (See also Step IV.2 below and About Creating and Applying Motion Frames to Imported Geometries (p. 533).)

2. From the **File** menu, **Write** the case to a location of your choosing. Tip: Remember the location of this directory as you will need to access files saved to it in later steps.

3. (Optional) Configure the options for Rocky Export by doing any or all of the following:

   - Define the directory you want to export by doing the following:

     a. From the Fluent **Rocky Export** menu, point to **Configure one-way export** and then click **Select Directory to Export**.

     b. From the **Select folder to write fluent to rocky files**, select the folder to where you want your fluid data saved, and then click **Select Folder**. Note: If you do not define this, Fluent will automatically use the same folder as your case file.

   - Define the additional variables you want to export by doing the following: Note: These additional variables can include static pressure, species variables such as mole fraction, diffusivity, and user defined memories. Other variables such as Velocity, Pressure Gradient, Temperature, and Fluid properties are always exported no matter what you select here.

     a. From the Fluent **Rocky Export** menu, point to **Configure one-way export** and then click **Select Variables to Export**. Note: To do this task, you must first have variables defined within Fluent.

     b. From the **Additional Variables Selector** dialog that appears, select the variables that you want included in the exported files, and then click **OK**.

   - Define how frequently you want files to be saved during the export by doing the following:

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

a. From the Fluent **Rocky Export** menu, point to **Configure one-way export** and then click **Set Output Frequency**.

b. From the **Fluent to Rocky (F2R) time-step multiplier** dialog define what you want for the **F2R Time Step Multiplier** value, and then click **OK**. Note: Only whole numbers are valid multipliers.

4. Export the data you want by doing one of the following:

   - To export steady-state data, do the following:

     a. Process the CFD simulation until the fluid flow reaches a steady state.

     b. From the Fluent **Rocky Export** menu, point to **Export one-way data** and then click **Export current data to Rocky**. Tips:

        – If you do not see this menu in Fluent, then first follow the Install Ansys Coupling Components (p. 792) procedure before continuing.

        – To perform this step before the Fluent case is done processing, stop Fluent from processing the case further before using **Rocky Export** menu.

          An F2R and other files needed for coupling is saved to the same folder as your Fluent case file (or the alternate folder you defined in step 3).

   - To export transient data, do the following:

     a. Begin the export process by doing one of the following:

        – To start recording from the initial conditions (without any prior Fluent results), initialize the Fluent case.

        – To start recording some time after Fluent results have been collected, process the CFD simulation until the fluid flow reaches the point that you want to start recording data, and then stop Fluent from processing the case further.

     b. From the Fluent **Rocky Export** menu, point to **Record one-way transient data**, and then click **Start one-way transient export**. Tip: If you do not see this menu in Fluent, then first follow the Install Ansys Coupling Components (p. 792) procedure before continuing.

     c. Begin (or resume) processing in Fluent until the point you want to stop recording data. Stop Fluent from processing the case.

     d. From the Fluent **Rocky Export** menu, point to **Record one-way transient data**, and then click **Stop one-way transient export**. An F2R and other files needed for coupling is saved to the same folder as your Fluent case file (or the alternate folder you defined in step 3).

5. Click **OK** to close the confirmation message dialog.

6. Save the Fluent case again.

STEP III: SET UP THE DEM SIMULATION IN ROCKY

1. In Rocky, set up your simulation project as you normally would. (See also Set Simulation Parameters (p. 414).) Tips:

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

- If you want to use the exact same geometries as in the Fluent simulation, you may choose to import the CAS, CAS.H5, or CAS.GZ file you generated in Step II, which will include all available geometries as separate components.
- If you enabled Energy Equations in the Fluent case, then ensure you also enable Thermal Modeling settings in Rocky. (See also Enable Thermal Modeling Calculations (p. 400).)

STEP IV: APPLY 1-WAY FLUENT COUPLING

1. From the **Data** panel, click **CFD Coupling** and then from the **Data Editors** panel, choose **Fluent (Fluid → Particle)** from the **Coupling Mode** list.
2. From the **Select Fluent 2 Rocky export file** dialog, locate and select the F2R file you exported in Step II, and then click **Open**. A new **1-Way Fluent** item appears beneath **CFD Coupling** in the **Data** panel. Important:For F2R files that include transient data, this step could take several minutes to complete. In addition, if you chose to set up Moving Meshes in Fluent, one new Frame will appear under **Motion Frames** in the **Data** panel for each Cell Zone with motions you have defined. Important:If you have defined your Moving Meshes in Fluent using expressions, Rocky will not be able to create the associated Motion Frames for you automatically. Rather, you must manually create your own Motion Frames in Rocky to match the Fluent motions as closely as possible. Tip: To save loading time, the Nodes for the imported fluid are turned off by default in the 3D View. To see them, from the Coloring tab, enable the **Nodes** checkbox. (See also About Using the Coloring Tab to Change a 3D View.)
3. Under **CFD Coupling**, select the new item and then from the **Data Editors** panel, on the **1-Way Fluent** tab, do the following:
   a. Set the **Start Time** you want.
   b. From the **Interactions** tab, do the following:
      i. From the **Particle** list, select (or multi-select) the Particle set name(s) for which you want to define interactions, and then select the various CFD laws you want for the selected set(s).
      ii. Select what you want for **Turbulent Dispersion**.

(See also About Using the 1-Way Fluent Method (p. 775).)

STEP IV: PROCESS THE COUPLED SIMULATION

1. In Rocky, process the simulation as you normally would. (See also Start Processing a Simulation from the Beginning (p. 913).)

See Also:

- About Using the 1-Way Fluent Method (p. 775)
- Set Up and Run a 1-Way Fluent Transient Project within Ansys Workbench (p. 789)
- Use a Journal to Run Fluent and Export Transient 1-Way Results (p. 1313)

USE THE 2-WAY FLUENT METHOD TO PROCESS A FLUENT AND ROCKY SIMULATION

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

There are five main steps to using the 2-Way Fluent method to process a coupled Rocky and Ansys Fluent simulation, as described below.

Note: Both Single phase (p. 1395) and Multiphase (p. 1392) approaches are covered in this procedure.

Step I: Verify that All Rocky-Fluent Program Requirements are Met

1. Before beginning Step II below, ensure all of the following program requirements have been met:

   - A Rocky-supported version of Ansys Fluent is installed upon the same machine that contains your Rocky program installation. (See also System Requirements (p. 67).)
   - If you desire to run Fluent as a parallel process (rather than a serial one), then the machine upon which you intend to run the Rocky portion of your coupled simulation must also run at least the host part of the Fluent process. (For more information about how the processing architecture works in Fluent, refer to your Ansys Fluent documentation.)
   - If you are not certain whether you have installed the **Ansys Fluent Coupling Support** component during Rocky installation, first follow the Install Ansys Coupling Components (p. 792) procedure before beginning Step II.

Step II: Set Up the Initial CFD Simulation

1. In Ansys Fluent, set up the CFD simulation that you want to couple with Rocky. Important:Ensure that you complete all of the following; these exact settings are required for 2-Way Fluent coupled simulations:

   a. From the **Fluent Launcher** dialog, do the following:

      i. Under **Dimension** ensure **3D** is selected.

      ii. Under **Options** select **Double Precision**.

   b. From the **General** tab, under **Time** select **Transient**.

   c. From the **Tree** panel, expand **Models**, and then do one of the following:

      - To set up a CFD project with a single fluid phase (p. 1395), ensure that **Multiphase** model is set to **Off**.
      - Otherwise, to set up a CFD project with two or more fluid phases (p. 1392), do all of the following:

        i. Under **Models**, select **Eulerian**. Note: The **Volume of Fluid** model is not supported by Rocky in this version.

        ii. Under **Eulerian Parameters**, keep all options cleared. (All are unsupported with coupling, including the **Multi-Fluid VOF Model**.)

        iii. Under **Volume Fraction Parameters**, select **Implicit** for **Formulation**.

        iv. Under **Number of Eulerian Phases**, select the number of phases that includes an additional one for particles. (The number of Eulerian phases should be set to number of fluid phases + 1.).

        v. Click **OK**.

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

d. Only for projects where the **Multiphase** model has been set to **Eulerian** (i.e., multiphase (p. 1392) simulations), ensure that you set the additional particle phase to have a different material from the primary phase.

e. Only for projects involving multiple fluid species, for the **mixture-template** for your **Material Mixture**, ensure that you have enabled **Species Transport**. Important:The species you list last in the **Selected Species** list is very important as Fluent considers it to be the bulk species; this *last species*, therefore, will not result in a mass source term from Rocky.

f. From the **Solution Methods** tab, under **Pressure-Velocity Coupling Scheme**, do one of the following:

   - For projects where the **Multiphase** model has been set to **Eulerian** (i.e., multiphase (p. 1392) simulations), you must ensure that the **Phase Coupled SIMPLE** method is selected.
   - For projects where the **Multiphase** model has been set to **Off** (i.e., single phase (p. 1395) simulations), you may choose any pressure-velocity coupling scheme that works for your particular application.

g. From the **Solution Methods** tab, under **Transient Formulation**, ensure that **First Order Implicit** is selected.

h. From the **Run Calculation** tab, under **Time Advancement**, do the following:

   i. Set **Type** to **Fixed**.

   ii. Ensure the **Time Step Size** value you set matches the minimum initial Output Frequency you want for your Rocky and Fluent files. Tip: You can change the outputs to be less frequent than this minimum in your Rocky setup later.

Tip:

To help ensure that you perform these setup steps correctly, please refer to the CFD Coupling Technical Manual document (from the **Help** menu, point to **Manuals**, and then click **CFD Coupling Technical Manual**). For single-phase approaches, you may also follow the self-guided:

Tutorial 14 - DEM-CFD Two-Way Coupling with Ansys Fluent.

Optional Steps:

- If you want to start your Rocky simulation with a given initial fluid flow, save the solution in Fluent as a standard DAT (DAT.H5 or DAT.GZ) file to be used by Rocky later (see Step IV below). Note: If your simulation is multiphase, ensure that the secondary phase (particulate phase) has a volume fraction of zero set before saving your DAT (DAT.H5 or DAT.GZ) file. Later, after the simulation is initialized, the initial particulate volume fraction will be calculated in Rocky according to the particles enabled at the beginning of the simulation.
- If you want to use any standard Fluent post processing options, set them up now before processing the simulation.

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

- If you want both sides of your coupled simulation to share identical geometries, set up those geometries now in Fluent, but be sure you save your Fluent Case File. Later in Rocky, you can choose to import those geometries directly from the CAS, CAS.H5, or CAS.GZ file that you will save from Fluent in the next step.

- If you want both sides of your coupled simulation to share the same geometry movements, set up the translation or rotation movements now using Fluent Moving Meshes. These movements will later be imported (with the CAS, CAS.H5, or CAS.GZ file) into Rocky (Step IV.2 below), which will automatically convert them into Rocky Motion Frames. Note: If you chose to define your moving meshes in Fluent using expressions, you will need to set up your Motion Frames in Rocky manually. (See also Step IV.2 below and About Creating and Applying Motion Frames to Imported Geometries (p. 533).)

1. Write the completed CFD setup to a CAS (CAS.H5 or CAS.GZ) file.

Step III: Set up the Initial Rocky Simulation

1. In Rocky, set up your simulation project as you normally would. (See also Set Simulation Parameters (p. 414).) Tip: If you want to use the exact same geometries as in the Fluent simulation, you may choose to import the CAS, CAS.H5, or CAS.GZ file you generated in Step II, which will include all available geometries as separate components.

2. Recommended: Save your Rocky project to the same folder in which you saved the CAS (CAS.H5 or CAS.GZ) file in Step II. (See also Save a New Simulation Project (p. 853).)

   Whether or not you choose to take this step, Rocky will create its own copy of the CAS (CAS.H5 or CAS.GZ).

3. Optional: If you want to start your coupled simulation with particle data already in the system, do all of the following:

   a. Process the simulation as you normally would, without CFD coupling. (See also Start Processing a Simulation from the Beginning (p. 913).)

   b. After processing is complete, from the **File** menu, click **Save project as…**.

   c. From the **Save As** dialog, choose the third option, "**Save as a New Project for Restart**", and then click **OK**.

   d. In the **Save in** list, choose the same location in which you saved the CAS (CAS.H5 or CAS.GZ) file in Step II, verify the **File Name**, and then click **Save**.

Step IV: Apply 2-Way Coupling

1. From the **Data** panel, click **CFD Coupling** and then from the **Data Editors** panel, choose **2-Way Fluent** from the **Coupling Mode** list.

2. From the **Select Fluent CAS file** dialog, locate and select the CAS (CAS.H5 or CAS.GZ) file you generated in Step II, and then click **Open**. Important:A mesh validation step will occur immediately after the CAS file import. This requires a valid Fluent license on the same machine upon which you are running the Rocky simulation. After successful import, a new **2-Way Fluent** item appears beneath **CFD Coupling** in the **Data** panel. In addition, if you chose to set up Moving Meshes in Fluent, one new Frame will appear under **Motion Frames** in the **Data** panel for each Cell Zone with motions you have defined. Important:If you have defined your Moving Meshes in Fluent using expressions, Rocky will not be able to create the associated Motion Frames for you automatically. Rather, you

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

must manually create your own Motion Frames in Rocky to match the Fluent motions as closely as possible.

3. If the CAS (CAS.H5 or CAS.GZ) file imported new Motion Frames, assign each newly imported Frame to the geometry component you want to share corresponding movements with Fluent. (See also Apply a Motion Frame to an Imported Geometry (p. 569).)

4. From the **Data** panel under **CFD Coupling**, select the new **2-Way Fluent** item and then from the **Data Editors** panel, do all of the following:

   a. From the **Interactions** tab, define the CFD laws you want per *Particle set* (p. 1393) by selecting (or multi-selecting) the set name(s) from the **Particle** list, and then defining the laws you want for the selected set(s). From this tab, you can also set the **Turbulent Dispersion** options for the entire coupled simulation. (See also About Using the 2-Way Fluent Method (p. 795).)

   b. From the **Coupling** tab, define the **Mapping Method** and **Sub-Stepping** options you want.

   c. From the **Zones and Interfaces** tab, do the following:

      - Select one or more options from the **Coupling Fluid Cell Zone** list.
      - (Optional) If provided, select the options you want from the **Mapping Cell Zone Interfaces** list.

   d. From the **Fluent** tab, do all of the following:

      i. **Important:**

      If your Fluent case is multiphase, from the **Rocky Phase** list, ensure that the phase representing particle flow is chosen.

      ii. Choose what fluid flow data Rocky starts with by doing one of the following:

      - To start your Rocky simulation without providing initial fluid flow data, ensure the **Use Data Initialization** checkbox remains cleared. Doing so ensures that the initialization settings prescribed in Fluent will be used to generate the initial fluid flow.
      - To start your Rocky simulation with initial fluid flow data, select **Use Data Initialization**, click the **Import File** button, and then select and open the DAT (DAT.H5 or DAT.GZ) file you saved in Step II. Important:If you chose to use initial fluid flow data in your simulation and your simulation is multiphase (p. 1392), you must have ensured that the secondary phase (particulate phase) had a volume fraction of zero set before you saved your DAT (DAT.H5 or DAT.GZ) file. This is important because later, after the simulation is initialized, the initial particulate volume fraction will be calculated in Rocky according to the particles enabled at the beginning of the simulation.

      iii. From the **Version** list, select what Ansys version you want to use for fluid coupling. Tip: If you do not see the Ansys version you expected, see also Rocky does not list my Ansys version when I try to set up 2-Way Fluent Coupling (p. 1326).

      iv. Choose how the fluid portion of the simulation is processed by doing one of the following:

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

- To have it processed on only one processor on your local machine, from the **Execution mode** list, select **Serial**.
- To have it processed on multiple processors on your local machine, from the **Execution mode** list, select **Local Parallel** and then from the **Solver Processes** field, enter the number of processors you want to dedicate to the CFD solver.
- To have it processed across several machines on your network, from the **Execution mode** list, select **Distributed Parallel** and then do one of the following:
  - Under **Hosts**, click the **Add** button for each server you want to add and then for each row, define the **Host name** and **Amount** information.
  - To import an external list of hosts and amounts that you created outside of Rocky, click the **Import File** button, and then from the **Import** dialog, locate and select the TXT file containing the information, and then click **Open**. The **Host** section is populated with the information from the TXT file. Tip: The format for the TXT file needs to include the **Host name** repeated on separate lines for each **Amount** you want devoted. To see an example, create a few sample rows as specified directly above, click **Export File** and then save and view the results as a TXT file.

v. Choose how many Fluent files to keep by doing one of the following:

- To keep each Fluent DAT (DAT.H5 or DAT.GZ) file (and in some cases, as with moving meshes, CAS files as well) for every Rocky output file that is saved, enable the **Keep all files** checkbox. Important:Unless you require all files for post-processing in Fluent, keeping all files is not recommended for the 2-Way Fluent method. (See also About Using the 2-Way Fluent Method (p. 795).)
- To keep only the last (or last several) Fluent DAT (DAT.H5 or DAT.GZ) files (and in some cases, CAS files) saved, ensure the **Keep all files** checkbox is cleared, and then enter the number of last-saved files you want to keep in the **Files to keep** box. Tip: Keeping the last 2 (or more) files is recommended. Important:If you decide to save fewer than all of the files, be aware that you will not be able to post-process the unsaved files in Fluent. Also, be aware that the number DAT (and in some cases, CAS) files you save will be as far back in time as you can restart your simulation since fluid data will not be available for prior times. Tip: You can also choose which quantities and frequency you want to write to disk using Fluent's Autosave functionality.

e. From the **Variables** tab, review the **Additional Inputs** and **Additional Outputs** (if applicable) that Rocky and Fluent will exchange during the coupled simulation. Tip: If you have enabled species transport, this is where the per-species variables Fluent provides Rocky, and the mass source terms Rocky provides Fluent will be listed.

Tips:

- To verify or change the CAS (CAS.H5 or CAS.GZ) file selected, click **Open** (at the bottom of the **2-Way Fluent** | **Fluent** tab) and Rocky will open its own copy of the CAS (CAS.H5 or CAS.GZ) file in Fluent. From here, you can view or make changes to the Rocky copy of the file. Important:Because Rocky uses only its own copy of the imported CAS (CAS.H5 or CAS.GZ) file, it is critical that you only open the imported CAS (CAS.H5 or CAS.GZ) from Rocky's **Open** button, and that you save any changes to the default location Rocky indicates.

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

- If you make changes to the Rocky copy of the CAS (CAS.H5 or CAS.GZ) file in Fluent, click **Refresh** to have Rocky include the updates. Important:The **Refresh** button works only if you first opened the CAS (CAS.H5 or CAS.GZ) file from the Rocky **Open** button located at the bottom of the **2-Way Fluent | Fluent** tab.

- If you made changes to your original Fluent version of the CAS (CAS.H5 or CAS.GZ) file and did not use the Rocky **Open** button as specified above, you must set the **CFD Coupling** option to **No Coupling** to clear the values, and then restart "Step IV: Apply 2-Way Coupling" again from the beginning, selecting the updated CAS (CAS.H5 or CAS.GZ) file in step 2.

Step V: Process the Coupled Simulation

In Rocky, process the simulation as you normally would (see also Start Processing a Simulation from the Beginning (p. 913)) with the following exception:

- Keeping in mind the **Time Step Size** value you set in Fluent, from the **Solver | Time** tab, set what you want for **Fluent Outputs Multiplier**. Tip: You can see how this change affects Rocky's Output Frequency in the disabled **Simulation** field.

Rocky automatically opens Ansys Fluent and the Fluent simulation is processed at the same time as the Rocky simulation, depending upon the limitations of your Ansys license. If any incompatibility is found between the Fluent setup and Rocky, an error will be shown in Rocky with additional information. Once the simulations are complete, the data from both are available to see and analyze however you choose. (See also Analyzing a Simulation (p. 931).)

See Also:

- About Using the 2-Way Fluent Method (p. 795)

USE THE 2-WAY FLUENT SEMI-RESOLVED METHOD TO PROCESS A FLUENT AND ROCKY SIMULATION

There are five main steps to using the 2-Way Fluent Semi-Resolved method to process a coupled Rocky and Ansys Fluent simulation, as described below.

Note: Both Single phase (p. 1395) and Multiphase (p. 1392) approaches are covered in this procedure.

Step I: Verify that All Rocky-Fluent Program Requirements are Met

1. Before beginning Step II below, ensure all of the following program requirements have been met:

    - A Rocky-supported version of Ansys Fluent is installed upon the same machine that contains your Rocky program installation. (See also System Requirements (p. 67).)

    - If you are not certain whether you have installed the **Ansys Fluent Coupling Support** component during Rocky installation, first follow the Install Ansys Coupling Components (p. 792) procedure before beginning Step II.

Step II: Set Up the Initial CFD Simulation

1. **Important**: When setting up your 3D mesh, ensure that only tetrahedron, prism, pyramid, or hexahedron cell types are used. Note: Polyhedral mesh cells, including the CutCell type, are not supported in this version.

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

2. In Ansys Fluent, set up the CFD simulation that you want to couple with Rocky. **Important**: Ensure that you complete all of the following; these exact settings are required for 2-Way Fluent Semi-Resolved

   coupled simulations:

   a. From the **Fluent Launcher** dialog, do the following:

      i. Under **Dimension** ensure **3D** is selected.

      ii. Under **Options** select **Double Precision**.

   b. From the **General** tab, under **Time** select **Transient**.

   c. From the **Tree** panel, expand **Models**, and then do one of the following:

      - To set up a CFD project with a single fluid phase (p. 1395), ensure that **Multiphase** model is set to **Off**.

      - Otherwise, to set up a CFD project with two or more fluidphases (p. 1392), do all of the following:

        i. Under **Models**, select **Eulerian**.

        ii. Under **Number of Eulerian Phases**, select the number of phases you need. Note: Unlike with the standard 2-Way Fluent method, you do not need an additional phase for particles with this method.

        iii. Click **OK**.

   d. From the **Solution Methods** tab, under **Pressure-Velocity Coupling Scheme**, you may choose any pressure-velocity coupling scheme that works for your particular application.

   e. From the **Solution Methods** tab, under **Transient Formulation**, ensure that **First Order Implicit** is selected.

   f. From the **Run Calculation** tab, under **Time Advancement**, do the following:

      i. Set **Type** to **Fixed**.

      ii. Ensure the **Time Step Size** value you set is aligned with the **Output Settings** | **Time Interval** value you want to set later in Rocky. Tip: To ensure that your post-processing is better aligned between the two products, it is recommended that the **Time Step Size** in Fluent and the **Output Settings** | **Time Interval** value in Rocky are multiples.

   Optional Steps:

   - If you want to start your Rocky simulation with a given initial fluid flow, save the solution in Fluent as a standard DAT (DAT.H5 or DAT.GZ) file to be used by Rocky later (see Step IV below).

   - If you want to use any standard Fluent post processing options, set them up now before processing the simulation.

   - If you want both sides of your coupled simulation to share identical geometries, set up those geometries now in Fluent, but be sure you save your Fluent Case File. Later in Rocky, you can

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

choose to import those geometries directly from the CAS, CAS.H5, or CAS.GZ file that you will save from Fluent in the next step.

- If you want both sides of your coupled simulation to share the same geometry movements, set up the translation or rotation movements now using Fluent Moving Meshes. These movements will later be imported (with the CAS, CAS.H5, or CAS.GZ file) into Rocky (Step IV.2 below), which will automatically convert them into Rocky Motion Frames. Note: If you chose to define your moving meshes in Fluent using expressions, you will need to set up your Motion Frames in Rocky manually. (See also Step IV.2 below and About Creating and Applying Motion Frames to Imported Geometries (p. 533).)

3. Write the completed CFD setup to a CAS (CAS.H5 or CAS.GZ) file.

Step III: Set up the Initial Rocky Simulation

1. In Rocky, set up your simulation project as you normally would. (See also Set Simulation Parameters (p. 414).)

   Tip:

   If you want to use the exact same geometries as in the Fluent simulation, you may choose to import the CAS, CAS.H5, or CAS.GZ file you generated in Step II, which will include all available geometries as separate components.

   Note:

   2-Way Fluent Semi-Resolved currently supports only the following particle shapes: Spheres, Shells (either rigid or flexible), or Custom Imported Polyhedrons (rigid only).

2. Recommended: Save your Rocky project to the same folder in which you saved the CAS (CAS.H5 or CAS.GZ) file in Step II. (See also Save a New Simulation Project (p. 853).) Whether or not you choose to take this step, Rocky will create its own copy of the CAS (CAS.H5 or CAS.GZ).

3. Optional: If you want to start your coupled simulation with particle data already in the system, do all of the following:

   a. Process the simulation as you normally would, without CFD coupling. (See also Start Processing a Simulation from the Beginning (p. 913).)

   b. After processing is complete, from the **File** menu, click **Save project as…**.

   c. From the **Save As** dialog, choose the third option, "**Save as a New Project for Restart**", and then click **OK**.

   d. In the **Save in** list, choose the same location in which you saved the CAS (CAS.H5 or CAS.GZ) file in Step II, verify the **File Name**, and then click **Save**.

Step IV: Apply 2-Way Fluent Semi-Resolved Coupling

1. From the **Data** panel, click **CFD Coupling** and then from the **Data Editors** panel, choose **Fluent Semi-Resolved** from the **Coupling Mode** list.

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

2. From the **Select Fluent CAS file** dialog, locate and select the CAS (CAS.H5 or CAS.GZ) file you generated in Step II, and then click **Open**. Important:A mesh validation step will occur immediately after the CAS file import. This requires a valid Fluent license on the same machine upon which you are running the Rocky simulation. After successful import, a new **2-Way Fluent Semi-Resolved** item appears beneath **CFD Coupling** in the **Data** panel. In addition, if you chose to set up Moving Meshes in Fluent, one new Frame will appear under **Motion Frames** in the **Data** panel for each Cell Zone with motions you have defined. Important:If you have defined your Moving Meshes in Fluent using expressions, Rocky will not be able to create the associated Motion Frames for you automatically. Rather, you must manually create your own Motion Frames in Rocky to match the Fluent motions as closely as possible.

3. If the CAS (CAS.H5 or CAS.GZ) file imported new Motion Frames, assign each newly imported Frame to the geometry component you want to share corresponding movements with Fluent. (See also Apply a Motion Frame to an Imported Geometry (p. 569).)

4. From the **Data** panel under **CFD Coupling**, select the new **2-Way Fluent Semi-Resolved** item and then from the **Data Editors** panel, do all of the following:

   a. Choose what fluid flow data Rocky starts with by doing one of the following:

      - To start your Rocky simulation without providing initial fluid flow data, ensure the **Use Data Initialization** checkbox remains cleared. Doing so ensures that the initialization settings prescribed in Fluent will be used to generate the initial fluid flow.

      - To start your Rocky simulation with initial fluid flow data, select **Use Data Initialization**, click the **Import File** button, and then select and open the DAT (DAT.H5 or DAT.GZ) file you saved in Step II.

   b. From the **Version** list, select what Ansys version you want to use for fluid coupling. Tip: If you do not see the Ansys version you expected, see also Rocky does not list my Ansys version when I try to set up 2-Way Fluent Coupling (p. 1326).

   c. Choose how the fluid portion of the simulation is processed by doing one of the following:

      - To have it processed on only one processor on your local machine, from the **Execution mode** list, select **Serial**.

      - To have it processed on multiple processors on your local machine, from the **Execution mode** list, select **Local Parallel** and then from the **Solver Processes** field, enter the number of processors you want to dedicate to the CFD solver.

      - To have it processed across several machines on your network, from the **Execution mode** list, select **Distributed Parallel** and then do one of the following:

        – Under **Hosts**, click the **Add** button for each server you want to add and then for each row, define the **Host name** and **Amount** information.

        – To import an external list of hosts and amounts that you created outside of Rocky, click the **Import File** button, and then from the **Import** dialog, locate and select the TXT file containing the information, and then click **Open**. The **Host** section is populated with the information from the TXT file. Tip: The format for the TXT file needs to include the **Host name** repeated on separate lines for each **Amount** you want devoted. To see an example, create a few

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

sample rows as specified directly above, click **Export File** and then save and view the results as a TXT file.

d. Choose how many Fluent files to keep by doing one of the following:

- To keep each Fluent DAT (DAT.H5 or DAT.GZ) file (and in some cases, as with moving meshes, CAS files as well) saved, enable the **Keep all files** checkbox. Important:Unless you require all files for post-processing in Fluent, keeping all files is not recommended for the 2-Way Fluent Semi-Resolved method. (See also About Using the 2-Way Fluent Semi-Resolved Method (p. 831).)

- To keep only the last (or last several) Fluent DAT (DAT.H5 or DAT.GZ) files (and in some cases, CAS files) saved, ensure the **Keep all files** checkbox is cleared, and then enter the number of last-saved files you want to keep in the **Files to keep** box. Tip: Keeping the last 2 (or more) files is recommended. Important:If you decide to save fewer than all of the files, be aware that you will not be able to post-process the unsaved files in Fluent. Also, be aware that the number of DAT (and in some cases, CAS) files you save will be as far back in time as you can restart your simulation since fluid data will not be available for prior times. Tip: You can also choose which quantities and frequency you want to write to disk using Fluent's Autosave functionality.

Tips:

- To verify or change the CAS (CAS.H5 or CAS.GZ) file selected, click **Open** and Rocky will open its own copy of the CAS (CAS.H5 or CAS.GZ) file in Fluent. From here, you can view or make changes to the Rocky copy of the file. Important:Because Rocky uses only its own copy of the imported CAS (CAS.H5 or CAS.GZ) file, it is critical that you only open the imported CAS (CAS.H5 or CAS.GZ) from Rocky's **Open** button, and that you save any changes to the default location Rocky indicates.

- If you make changes to the Rocky copy of the CAS (CAS.H5 or CAS.GZ) file in Fluent, click **Refresh** to have Rocky include the updates. Important:The **Refresh** button works only if you first opened the CAS (CAS.H5 or CAS.GZ) file from the Rocky **Open** button.

- If you made changes to your original Fluent version of the CAS (CAS.H5 or CAS.GZ) file and did not use the Rocky **Open** button as specified above, you must set the **CFD Coupling** option to **No Coupling** to clear the values, and then restart "Step IV: Apply 2-Way Fluent Semi-Resolved Coupling" again from the beginning, selecting the updated CAS (CAS.H5 or CAS.GZ) file in step 2.

Step V: Process the Coupled Simulation

In Rocky, process the simulation as you normally would (see also Start Processing a Simulation from the Beginning (p. 913)) with the following exception:

- Keeping in mind the **Time Step Size** value you set in Fluent, from the **Solver** | **Time** tab, set what you want for **Output Settings** | **Time Interval**. Tip: To ensure that your post-processing is better aligned between the two products, it is recommended that the **Time Step Size** in Fluent and the **Output Settings** | **Time Interval** value in Rocky are multiples.

Rocky automatically opens Ansys Fluent and the Fluent simulation is processed at the same time as the Rocky simulation, depending upon the limitations of your Ansys license. If any incompatibility is found between the Fluent setup and Rocky, an error will be shown in Rocky with additional information. Once the simulations are complete, the data from both are available to see and analyze however you choose. (See also Analyzing a Simulation (p. 931).)

See Also:

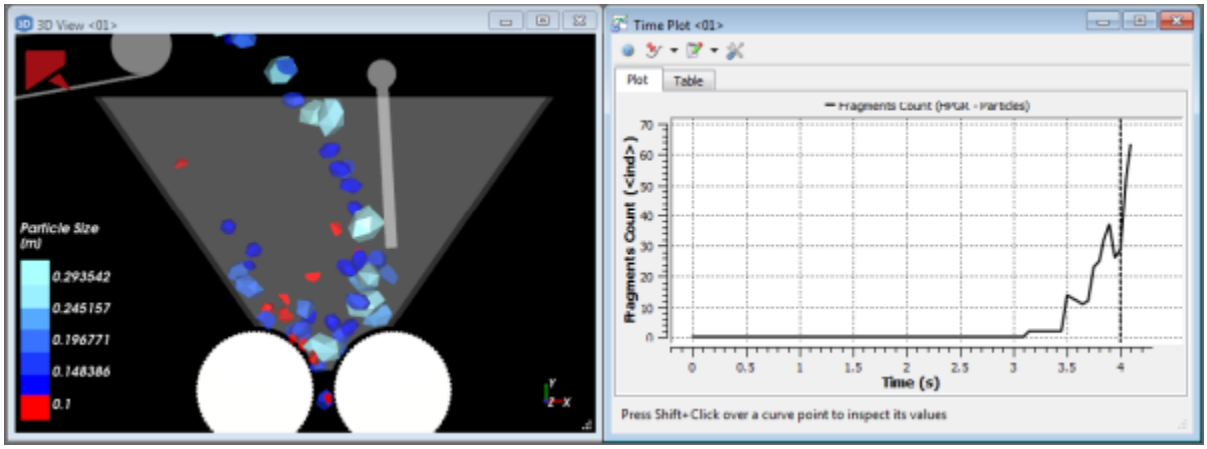
Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

- About Using the 2-Way Fluent Semi-Resolved Method (p. 831)

ENABLE AND VIEW PARTICLE BREAKAGE

Analyzing either Instantaneous (p. 1389) or Discrete (p. 1392) particle breakage in Rocky involves first turning on breakage parameters for your *Particle set* prior to processing your simulation (see also About Particle Breakage (p. 665)), and then viewing the results and/or particle-fragment-related Properties or Curves in a 3D View window or graphing them in a plot or histogram window (Figure 1).

Figure 2.227: Example of Instantaneous Breakage being simulated in an HPGR machine



It is important to realize that breakage data is only collected after the Breakage **Start** value on the **Solver** | **Time** tab is reached. This is to allow enough time for the particle flow within your simulation to reach a steady state before breakage calculations begin.

For further details on the specific models, tips, and limitations see the About Particle Breakage (p. 665) topic.

2.8.1. To Enable and View Particle Breakage:

1. Set up your simulation as you normally would (see also Set Simulation Parameters (p. 414)), ensuring all of the following:

   a. During the Set Simulation-Wide Parameters (p. 414) step, on the **Solver** | **Time** tab, you have set the values you want for both Breakage **Start** and Breakage **Delay after Release**. (See also About Solver Parameters (p. 464).) Tip: It is best practice to set your Breakage **Start** time to begin after a steady state has been reached for your particle flow.

   b. During the Add and Edit Particle Sets (p. 602) step, you have done all of the following for each of the |Gloss7|s you want breakage enabled:

      i. On the **Particle** tab, you have ensured that one of the following **Shape** | **Composition** option combinations are selected:

         - Either Straight or Custom *Fiber* Shape | Multiple Element Composition
         - Custom *Shell* Shape | Multiple Element Composition
         - *Solid* Polyhedron Shape | either Single Element or Multiple Element Composition

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

- Solid Briquette Shape | Single Element Composition
- Solid Faceted Cylinder Shape | Single Element Composition
- Solid Custom Convex Polyhedron Shape | either Single Element or Multiple Element Composition
- Solid Custom Concave Polyhedron Shape| Multiple Element Composition

ii. If you have selected a **Single Element** particle set, you have done all of the following:

A. On the **Particle** | **Breakage** tab, you have selected the **Enable Breakage** checkbox.

B. From the **Criteria** sub-tab, you have selected the *Instantaneous Breakage* model you want from the **Model** list, and have defined its associated parameters.

C. From the **Fragments** sub-tab, you have entered the values you want under **Limits**, have chosen the model you want from the **Distribution model** list, and have defined its associated parameters.

iii. If you have selected a **Multiple Element** particle set, you have done all of the following:

A. On the **Particle** | **Breakage** tab, you have selected the *Discrete Breakage* model you want from the **Model** list.

B. You have defined the model's associated parameters.

2. Process your simulation as you normally would. (See also About Starting a Simulation (p. 908).)
3. After your simulation reaches as least as far as the Breakage **Start** time, do one or more of the following:
   - From a 3D View window (see also About 3D View Windows (p. 1040)), observe the changes to the particle shapes as they fragment and break.
   - From the **Data** panel, select **Particles** and then from the **Data Editors** panel, select either the **Properties** or **Curves** tab and then create a plot (see also Graph Data within Rocky by Creating a Plot or Histogram (p. 1124)) or visualize the resulting fragments-related Properties in a 3D View window (see also About 3D View Windows (p. 1040)).

See Also:

- About Particle Breakage (p. 665)
- About Collecting Data in Rocky (p. 306)
- About Adding and Editing Particle Sets (p. 602)
- About Properties (p. 933)
- About Curves (p. 989)

CREATE AND VIEW A PARTICLE ASSEMBLY

(See also About Particle Assemblies (p. 663).)

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

To Create and View a Particle Assembly:

1. Begin setting up your simulation as you normally would (see also Set Simulation Parameters (p. 414)).
2. During the Add and Edit Particle Sets (p. 602) step, do the following:
    a. For each part (p. 1393) you want represented in your final Assembly (p. 1389) shape, create an individual Particle set (p. 1393). (See also Add a New Particle Set (p. 678).) Tip: Ensure your Particle set meets the specific criteria for Assembly parts. (Refer to the *Assembly Particle Shape Limitations* section of the Particle and Input Limitations (p. 59) topic for details.)
    b. Create another new Particle set (see also Add a New Particle Set (p. 678)), select it, and then from the **Data Editors** panel, do the following:
        i. From the main **Particle** tab, define the **Shape** as an **Assembly**.
        ii. From the **Shape** sub-tab, add a new definition row for each individual part you want to include, and define each's part's position, size scale, and rotation parameters. (See also About Adding and Editing Particle Sets (p. 602).) Tip: Check your settings in the **Particles Details** window. (About Particles Details Windows (p. 679).) Note: For Assembly shapes, the Particles Details 3D view shows the actual position of the part when the size type is Original Size Scale and it is set to 1 (which is the default setting).
    c. Define the rest of the shape settings as you normally would for any other Particle set. (Define what parameters you want on the **Size**, **Movement**, and **Orientation** sub-tabs.) Important:At this point, Rocky will calculate the mass, area, and other properties for the (whole) Assembly particle based on the values you have defined for the Particle sets upon which the individual parts are based, and the size of the parts you have defined within the Assembly. These properties will be calculated accurately by Rocky as long as your individual parts do not overlap; otherwise, the accuracy of these Rocky-calculated properties cannot be guaranteed and you should consider defining them yourself by specifying **Custom Properties**.
    d. (Optional) If your Assembly shape requires its parts to overlap, and you require accurate mass, area, and other properties for your Assembly particle, consider specifying **Custom Properties** for your assembly by doing the following:
        i. From the CAD program with which you designed your ideal (Assembly) particle shape, note the Area, Volume, Mass, Geometric Center, Center of Mass, and other properties you want Rocky to use for your Assembly particle shape.
        ii. In Rocky, from your Assembly particle's **Particle | Custom Properties** tab, enable the **Change Assembly Properties** checkbox, and then enter the information you want Rocky to use. Tip: You can use the colored dots in the **Particles Details** window to preview where your Center of Mass (blue dot) and Geometric Center (yellow dot) are located.
3. Finish setting up your simulation, process it, and then analyze the results as you normally would.

See Also:

- About Particle Assemblies (p. 663)
- About Adding and Editing Particle Sets (p. 602)
- Particle and Input Limitations (p. 59)

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

- About Defining and Importing Custom Particle Shapes (p. 670)

ENABLE THERMAL MODELING CALCULATIONS

If you want to observe how particles are affected by temperature changes brought on by the heating or cooling of other items around it, you may set up your simulation to model thermal properties. Turning on Thermal Modeling enables you to simulate conductive heat transfer from particles to other particles, and from particles to boundaries. When used with CFD Coupling methods, it can also simulate convective heat transfer between particles and fluids. (See also Set or Modify Fluid and/or Air Flow Properties (p. 757).)

Tip:

For examples and walk-through tutorials of thermal modeling in action, review the following resources:

- Tutorial - Conical Dryer in the *Rocky Tutorial Guide*
- Tutorial 13 - DEM-CFD One Way Coupling With Ansys Fluent (Workbench)

To enable Thermal Modeling in DEM calculations:

1. Set up your simulation as you normally would (see also Set Simulation Parameters (p. 414)), ensuring all of the following:

   a. During the Set Simulation-Wide Parameters (p. 414) step, on the **Physics** tab, select the **Enable Thermal** checkbox. (See also About Physics Parameters (p. 417).) Note: If you decide to enable Intra-particle Collision Statistics, know that the Temperature property will not be available to view on the **Particles Details** window, but can still be viewed from the main **Particles** entity or through a **3D View** window. (Se also About Intra-Particle Collision Statistics (p. 324).)

   b. During the Add and Edit Geometry Components (p. 480) step, on the **Geometry** sub-tab for each of your wall components, set what you want for **Thermal Boundary Type** and **Temperature**.

   c. During the Add and Edit Particle Sets (p. 602) step, on the **Particle** tab for the Particle set(s) you want affected by Thermal, ensure that on the **Composition** sub-tab for any particles you have you have selected **Multiple Elements** from the **Composition** list, that you also set the **Conductivity Ratio** parameter to the value you want.

   d. During the Modify Material Properties (p. 591) step, on the **Materials** tab for each of your particle and/or boundary materials, set all of the following parameters:

      - **Thermal Conductivity**
      - **Specific Heat**
      - **Poisson's ratio** (See also About Modifying Material Compositions (p. 591).)

   e. During the Add and Edit Particle Inputs (p. 708) step, for each Particle Input, set the **Temperature** of each Particle set row. (See also About Adding and Editing Particle Inputs (p. 708).)

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

f. If coupling your simulation with CFD methods, then during the Set or Modify Fluid and/or Air Flow Properties (p. 757) step, set what you want for the **Convective Heat Transfer Law**. (See also About Using the 1-Way Fluent Method (p. 775), About Using the 2-Way Fluent Method (p. 795), or About Using the 1-Way Constant Method (p. 763)). Note: If coupling your simulation with the 1-Way Constant method specifically, ensure you also set the values you want for constant **Temperature**, **Thermal Conductivity**, and **Specific Heat**. (See also About Using the 1-Way Constant Method (p. 763).)

2. Process your simulation as you normally would. (See also About Starting a Simulation (p. 908).)

To enable Thermal Modeling in SPH calculations:

1. Set up your simulation as you normally would (see also About SPH Parameters (p. 436)), ensuring all of the following:

   a. During the Set Simulation-Wide Parameters (p. 414) step, on the **Physics** tab, select the **Enable Thermal** checkbox. (See also About Physics Parameters (p. 417).) The Cleary model for thermal transfer is automatically selected.

   b. During the Add and Edit Geometry Components (p. 480) step, on the **Geometry** sub-tab for each of your wall components, set what you want for **Thermal Boundary Type** and **Temperature**.

   c. **If you simulation also involves DEM particles**: During the Add and Edit Particle Sets (p. 602) step, on the **Particle** tab for the Particle set(s) you want affected by Thermal, ensure that on the **Composition** sub-tab for any particles you have you have selected **Multiple Elements** from the **Composition** list, that you also set the **Conductivity Ratio** parameter to the value you want.

   d. During the Modify Material Properties (p. 591) step, on the **Materials** tab set all of the following parameters for the fluid material:

      - **Thermal Conductivity**
      - **Specific Heat**

2. Process your simulation as you normally would. (See also About Starting a Simulation (p. 908).)

Tip:

When you are ready to analyze your simulation (see also Analyzing a Simulation (p. 931)), you might pay special attention to the following Properties (see also About Properties (p. 933)):

- Temperature (for walls, conveyors, particles and fluids)
- Specific Heat (only for fluids)
- Thermal Conductivity (only for fluids)

See Also:

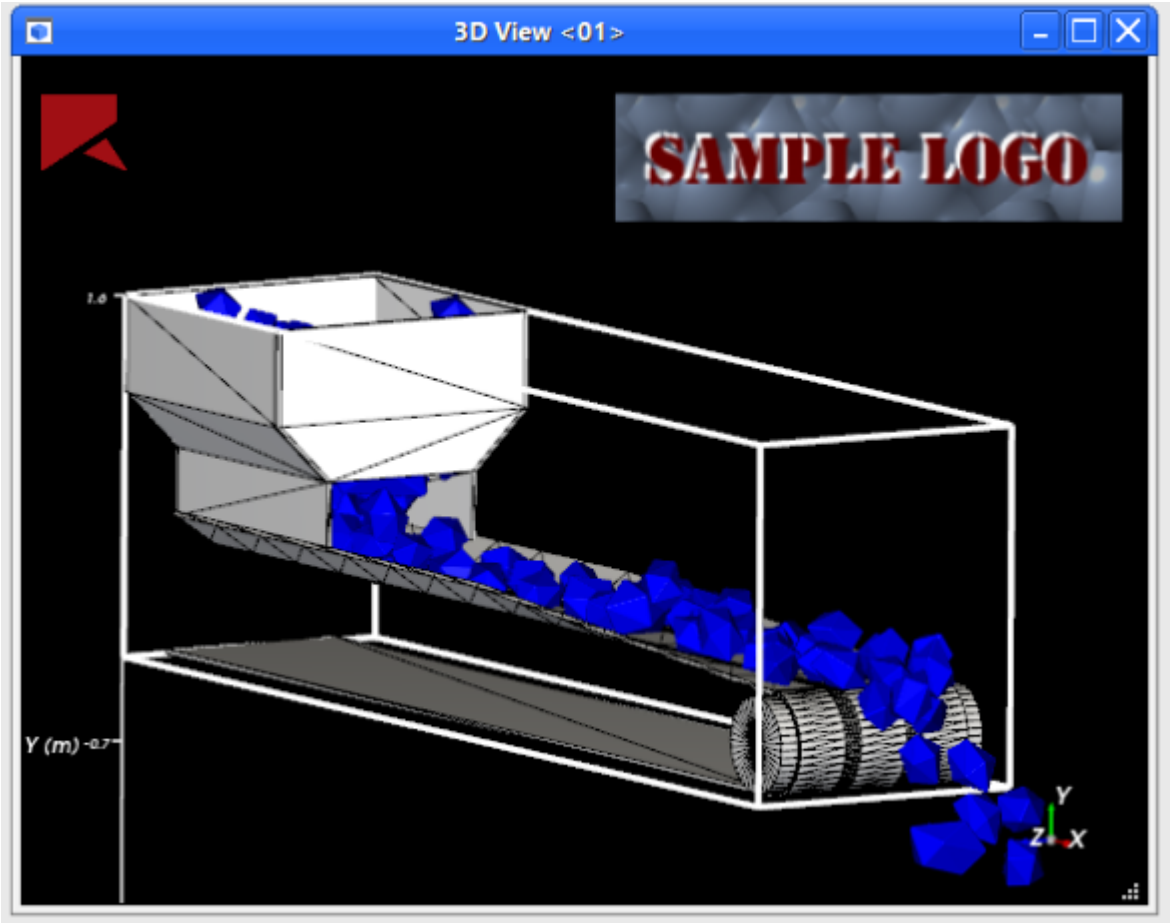
- View Frequently Used Tasks (p. 371)

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

ADD A LOGO TO A 3D VIEW

A 3D View of your simulation will always contain the Rocky logo but you can choose to add your own company logo to the view, too. This may be useful in cases where you are showing simulation images or animations to clients and want to brand your work.

Figure 2.228: 3D View window with a Sample Logo added as an Image Overlay



You add a logo image by using the Overlay tab on the **Window Editors** panel. Logos are added as images and can be positioned anywhere on the 3D View window that you like.

To add a logo to a 3D View:

1. Using separate image editing software, open the logo file you want to use and ensure the display size is appropriate.

2. Ensure your image is saved as either a PNG, JPG, BMP, or PNP file.

3. Open the simulation project to which you want to add the logo and select the 3D View window you want.

4. Using the **Overlay** tab on the **Window Editors** panel, add the logo to the 3D View as an Image Overlay. (See also Add or Edit an Image Overlay for a 3D View Window (p. 1070).)

See Also:

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

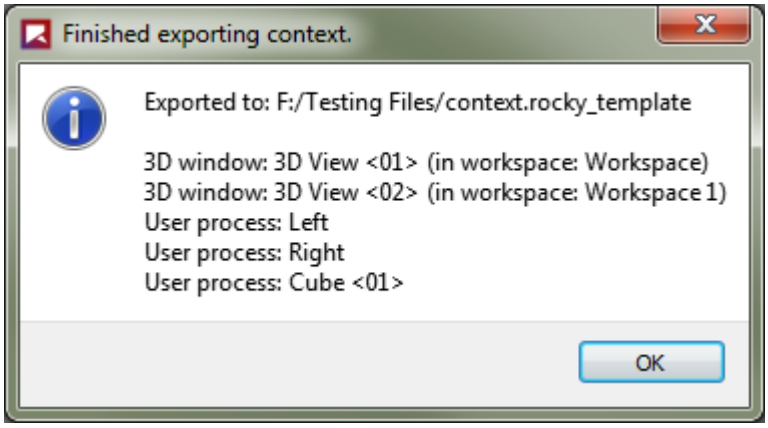
- About Using the Window Editors panel to Change a 3D View (p. 1058)
- View Frequently Used Tasks (p. 371)

REUSE 3D VIEW WINDOW, WORKSPACE, AND USER PROCESS SETUPS IN OTHER PROJECTS

For projects where you have spent time and effort setting up several 3D View windows, Workspaces (p. 1396), and User Processes just the way you like, you can now export all of that setup criteria and reuse it in other projects. This is an alternative to recording or writing a script (p. 1394) to do the setup for you, which requires you to know beforehand exactly how you want your views and processes to work. With this feature, you can decide after the fact whether you want to reuse the setup for another project or not.

This can be useful in cases where you have already run several similar simulations but your client suddenly wants to see new views or processes. By using this feature, you can set up the new views and processes in one project, and then apply those same views and processes to the rest of the projects in your set.

Rocky enables you to export this setup criteria to either a `rocky_template` file or to the clipboard, and even lists each component that was exported in a separate dialog (Figure 1) for confirmation purposes.



Example dialog that appears after exporting project context

In your new project, you can then choose to import the saved criteria from those same two locations.

Once imported, Rocky simply creates new 3D View windows, Workspaces, and User Processes to match what was exported. If you already have 3D View windows and User Processes in your project at the time of Import, Rocky will retain these items while adding the new ones to the project.

To export 3D View window, Workspace, and User Process setups for use in other projects:

1. Ensure that your 3D View windows, Workspaces, and User Processes are set up the way you want.
2. From the **File** menu, point to **Export project context** and then do one of the following:

- To ensure reuse of the project context for longer than just the immediate timeframe, select **To file**, and then do all of the following: a. From the **Save project context** dialog, select the location to which you want to save the `rocky_template` file, name it in the **File name** box, and then click **Save**. b. In the **Finished exporting context** dialog, review the items listed, and then click **OK**.

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

- To reuse the project context for only the immediate timeframe, select **To clipboard**, and then in the **Finished exporting context** dialog, review the items listed, and then click **OK**.

  The 3D Window, Workspace, and User Process setups are ready to import into another project.

To import 3D View window, Workspace, and User Process setups previously exported out of another project

1. Ensure the project that you want to apply the previously exported setups is open in Rocky.
2. From the **File** menu, point to **Import project context** and then do one of the following:

- To import from a previously exported file, select **From file**, and then from the **Restore project context** dialog, locate and select the `rocky_template` file you want to use, and then click **Open**.
- To import from the clipboard, select **From clipboard**.

  The 3D Window, Workspace, and User Process setups you previously exported are added to the project.

See Also:

- About Creating and Using PrePost Scripts (p. 296)
- Create and Modify a 3D View (p. 1040)
- Filter Views and Data with User Processes (p. 1174)
- About Workspaces (p. 108)
- View Frequently Used Tasks (p. 371)

SET UP A NEW GPU CARD

When setting up a new GPU card for use with Rocky, there are two steps you can take that might help make processing your simulations on GPU smoother and faster:

- Turn off the Windows Timeout Detection and Recovery (TDR)
- Enable double-precision processing

See below for specific instructions.

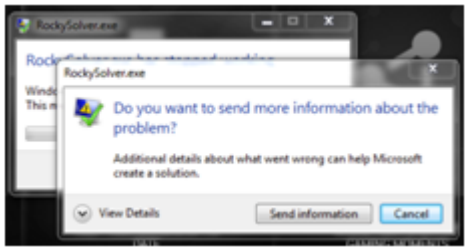
Turn off Timeout Detection and Recovery (TDR) for a GPU card

Application: This procedure is particularly useful when you have a single NVIDIA GPU card. But it can also be useful for any GPUs used by Rocky in general.

Symptom: Your Rocky simulations sporadically stop during the middle of a complex simulation, resulting in an incomplete calculation and an error (Figure 1). This is caused by Windows believing the GPU card is non-responsive because it is taking longer than expected to return an answer on a difficult calculation. Turning off TDR for your GPU allows the card the necessary time to execute the computation.

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

Figure 2.229: Error seen in Rocky when TDR causes the GPU to stop processing



To turn off TDR for a GPU card:

1. Ensure that you have successfully installed your GPU card and associated drivers, and have restarted your computer.

2. Download and install the latest NVIDIA Nsight Visual Studio Edition from Nvidia.

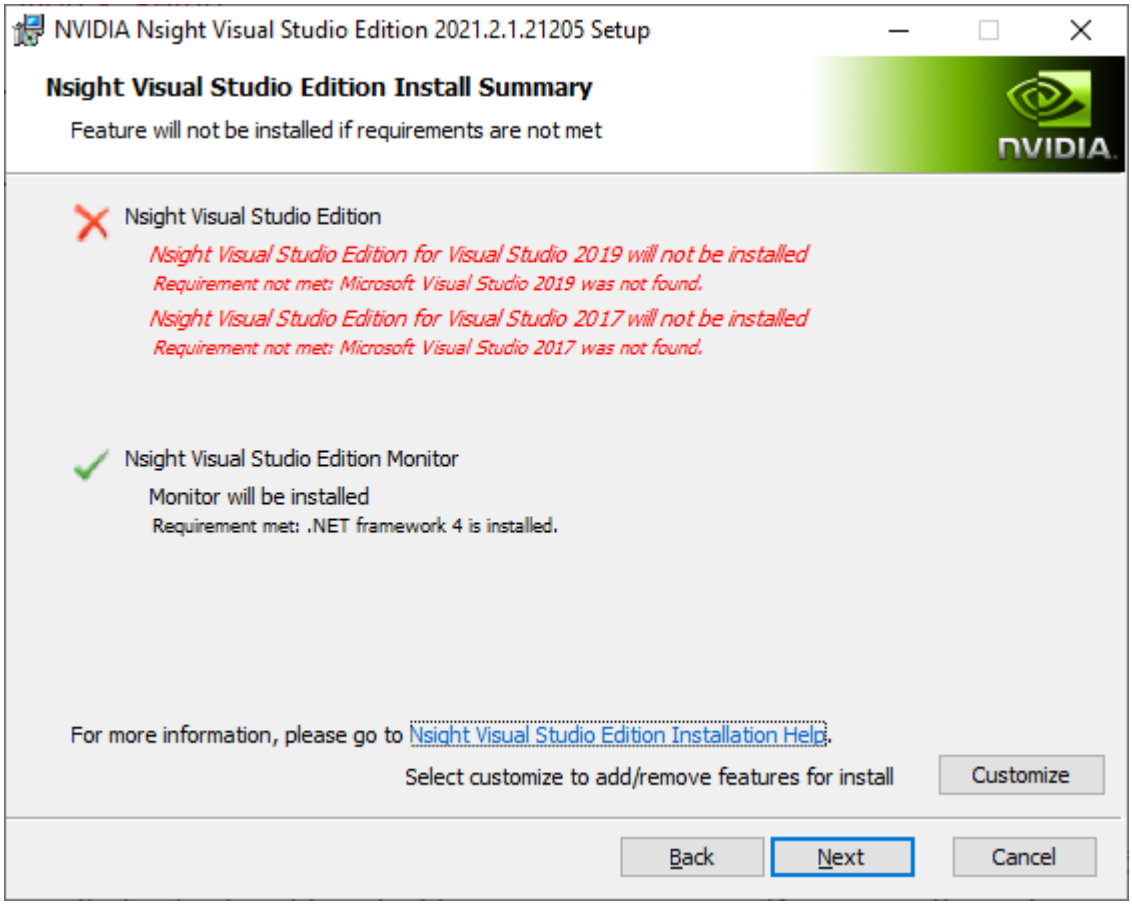


Figure 2: Installation confirmation screen for Nsight Visual Studio Edition

Notes:

- As seen in Figure 2 above, you will need only to install the Nsight Monitor option; the Nsight Visual Studio option is not necessary.

- You may need to Subscribe as a developer (free of charge) on the Nvidia website, if you haven't done it before, in order to download the Nsight Visual Studio Edition (Figure 3).

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

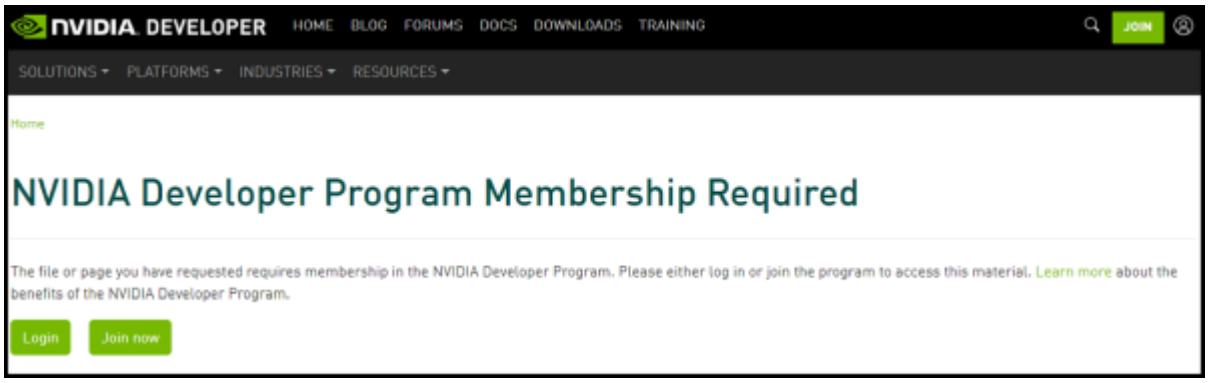


Figure 3: Subscribe to Nvidia developer program

3. Restart your computer.

4. From the Windows **Start** menu, under **NVIDIA Corporation**, click **Nsight Monitor** (Figure 4).

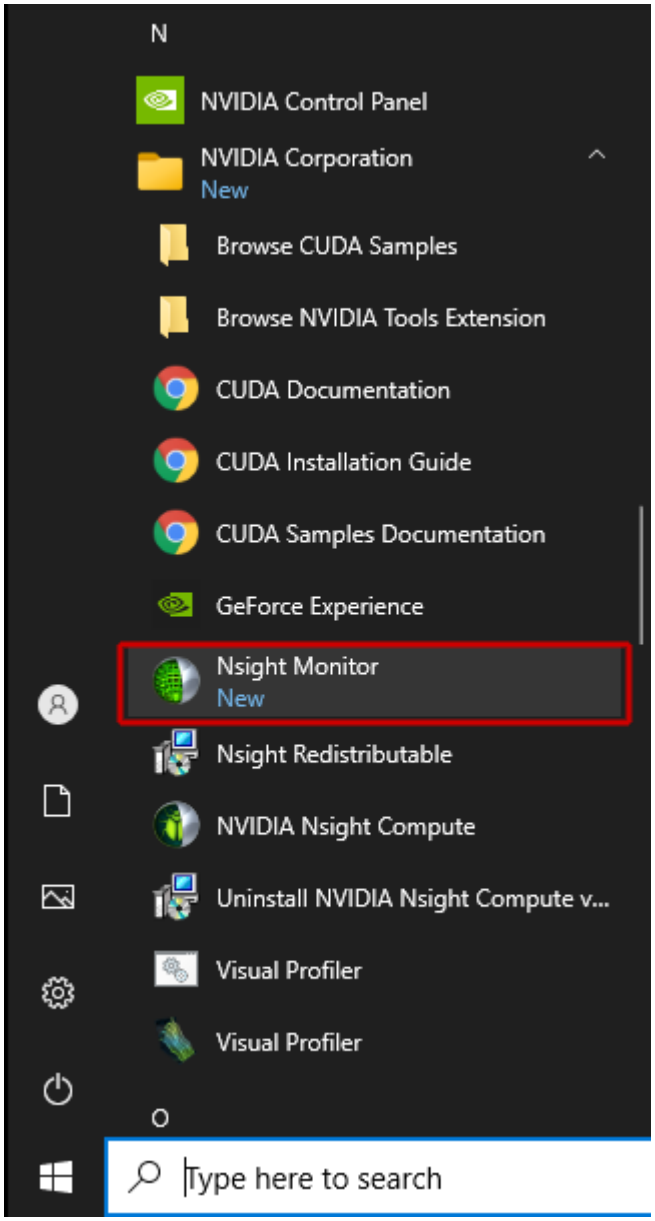


Figure 4 Nsight Monitor in Windows Start menu A new **Nsight Monitor** icon appears in your Notification Area.

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

5. From your Notification Area, right-click the **Nsight Monitor** icon and then select **Options** (Figure 5).

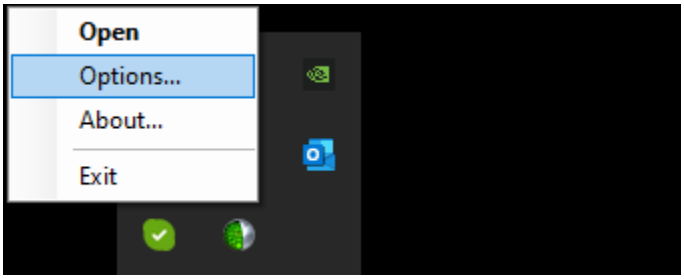


Figure 5: Nsight Monitor in Notification Area

6. From the **NVIDIA Nsight Options** dialog, on the **General** tab, set **WDDM TDR enabled** to **False** (Figure 6).

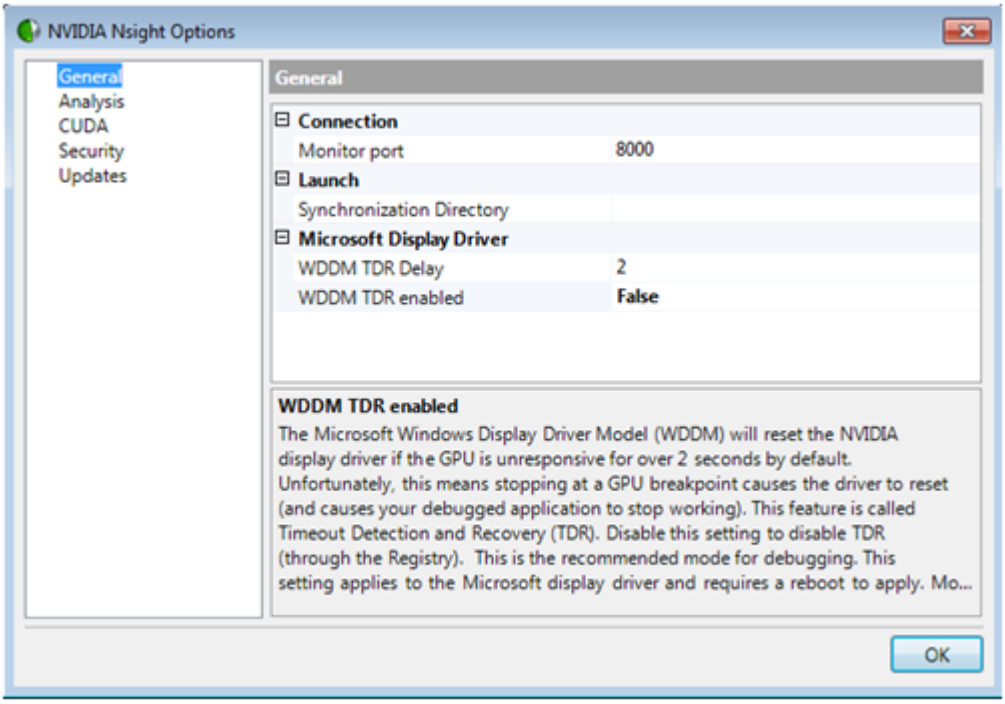


Figure 6: NVIDIA Nsight Options dialog

7. Click **OK**.

8. Restart your computer.

Enable double-precision processing for your GPU cards

Application: You have one or more GPU cards from the NVIDIA Titan family.

Symptom: You are seeing slower than expected results on your Titan family GPU card. This could be a result of not yet enabling double-precision processing. Doing so will speed up certain Rocky calculations, thereby reducing your overall processing time, at the expense of slowing down your graphics visualization performance. Additionally, it has been found that a Titan card using double precision will take longer when simulating Spheres (~20%), but will be faster when simulating shaped particles (p. 1395) (~200%).

To enable double-precision processing for your GPU cards:

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

1. Ensure that you have successfully installed your Titan card(s) and associated drivers, and have re-started your computer.

2. Right-click your Windows desktop, and then click **NVIDIA Control Panel** (Figure 7).

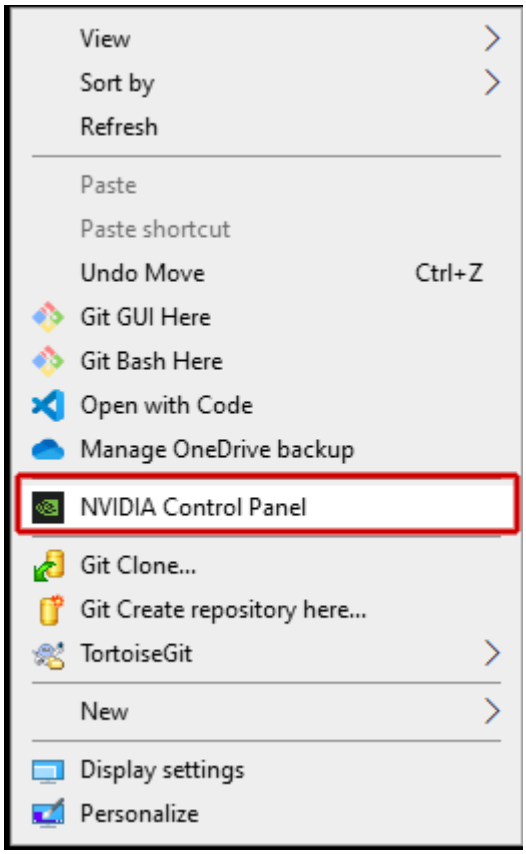


Figure 7: Windows desktop right-click menu showing NVIDIA Control Panel option

3. In the **NVIDIA Control Panel** window, do all of the following:

   a. From the **Select a Task** list, select **Manage 3D settings**.

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

b. In the **Global Settings** tab under **Feature**, select **Double precision** and then from the drop-down list, ensure all your Titan GPU cards are enabled (checked) (Figure 8).

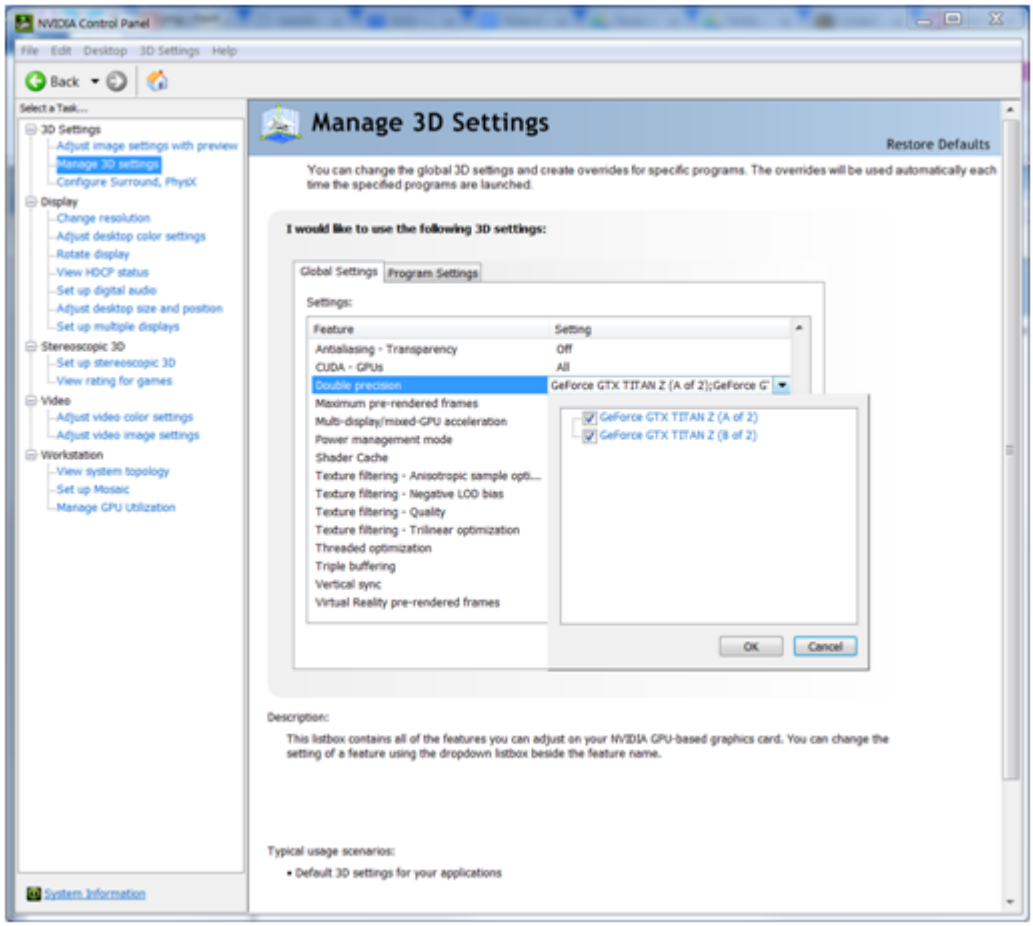


Figure 8: Manage 3D Settings section in the NVIDIA Control Panel

c. Click **OK**.

4. Restart your computer.

See Also:

- Best Practices for Faster Processing (p. 68)

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information
of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

Chapter 3: Setting Up a Simulation

Before you can process a simulation, you must first set the parameters upon which the simulation will be based. These parameters are set within the Data and Data Editors panels of the Rocky user interface. They include such settings as project name; particle type, size, and shape; geometry type and location; materials and interactions; CFD Coupling options; and more.

3.1. Creating a Simulation Project

Rocky opens by default with no new simulation project (p. 1394), so you must first choose what kind of project you want to open. You can begin a brand new (blank) project, open an existing simulation project, or restore an archived simulation project.

Note: No matter how many Rocky licenses you have, Rocky is designed to have only one instance of a simulation project opened at any one time. This means that whenever you open a project file in Rocky, that project automatically becomes "locked" to your instance of Rocky until you close it. This step helps prevent yourself and other users in your network from accidentally opening up a duplicate instance of the project and making changes. (See also Rocky says my project file is "locked" (p. 1382).)

See below for explanations on the various ways you can open a project in Rocky.

3.1.1. Minimum Setup Requirements

Rocky comes preset with default values that you can use right away without modification if you choose. However, the minimum requirements for processing a simulation include setting up each of the following items:

- One Particle set. (See also About Adding and Editing Particle Sets (p. 602).)
- One Input. (See also About Adding and Editing Particle Inputs (p. 708).) **Note**: If the input is a **Continuous Injection** type, then one inlet is also required. The inlet can either be from a default feed conveyor or from a separate inlet geometry that you have added into the simulation.

Tip: To see a walk-through example of setting up, processing, and analyzing a simple simulation, review Tutorial - Transfer Chute in the *Rocky Tutorial Guide*.

3.1.2. Open a Brand New Simulation Project

If you want to begin a new, blank simulation using no prior Rocky settings nor simulation results, you can choose to Begin a New Simulation Project (p. 412).

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

3.1.3. Open an Existing Simulation Project

To view or edit a project that was set up and/or processed in Rocky v3, Rocky v4 or later versions, you can Open an Existing v3 Simulation Project (p. 412) or Open an Existing v4 or Later Simulation Project (p. 413).

3.1.4. Restoring an Archived Simulation Project

If you have received an archived simulation project from someone else, or have archived your own project and have lost access to the original .rocky project, you can choose to Restore an Archived Simulation Project (p. 413). Restoring a project opens the simulation project from the .rocky_archive file. This is the same as opening the .rocky file directly so is only necessary in cases where you have only the .rocky_archive file and not the original .rocky simulation project file.

What do you want to do?

- Begin a New Simulation Project (p. 412)
- Open an Existing v3 Simulation Project (p. 412)
- Open an Existing v4 or Later Simulation Project (p. 413)
- Restore an Archived Simulation Project (p. 413)

See Also:

- Setting Up a Simulation (p. 411)

3.1.5. Begin a New Simulation Project

- From the **File** menu, click **New Project**. The **Data** panel is populated with default sections including Study, Physics, and Geometries.

See Also

- Open an Existing v3 Simulation Project (p. 412)
- Open an Existing v4 or Later Simulation Project (p. 413)

3.1.6. Open an Existing v3 Simulation Project

Note:

Opening projects that were processed in older versions of Rocky will have different features than those processed in Rocky v4 or later versions. To make use of newer Rocky features on older projects, you may need to re-set up and re-process it in the newer version of Rocky.

---

1. From the **File** menu, click **Open Project**.

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

2. Locate and select the .rocky30 file you want, and then click **Open**. The file opens with the settings, results, and analyses items you have saved.

See Also

- Begin a New Simulation Project (p. 412)

3.1.7. Open an Existing v4 or Later Simulation Project

1. From the **File** menu, click **Open Project**.
2. Locate and select the .rocky file you want, and then click **Open**. The file opens with the settings, results, and analyses items you have saved.

Notes:

- If you have a single-instance license of Rocky, you will be limited to opening only one instance of the Rocky program at a time. Attempting to open another project from a file directory, for example, might result in a "No license for feature" error on the License Management screen (see also My Rocky license will not let me open a project from the File directory (p. 1385)) so it is important to only use the above procedure to open Rocky projects when Rocky is already open.
- No matter how many Rocky licenses you have, Rocky is designed to have only one instance of a simulation project opened at any one time. This means that whenever you open a project file in Rocky, that project automatically becomes "locked" to your instance of Rocky until you close it. This step helps prevent yourself and other users in your network from accidentally opening up a duplicate instance of the project and making changes. (See also Rocky says my project file is "locked" (p. 1382).)

See Also

- Begin a New Simulation Project (p. 412)

3.1.8. Restore an Archived Simulation Project

1. From the **File** menu, click **Restore archived project**.
2. Locate and select the `.rocky_archive` file you want, and then click **Open**. The file opens the associated simulation project information, including the settings, results, and analyses items you have saved.

Tip:

You may also use a drag and drop method to restore an archived simulation project. From your file directory, click the `.rocky_archive` file you want to restore, drag it to an empty location on a Rocky Workspace (p. 1396) or the **Data** panel, and then when you see the **+ Copy** icon, drop the file by releasing your mouse.

See Also:

- Open a Simulation Project (p. 411)

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

- Archive a Simulation Project (p. 1294)

3.2. Setting up Simulation Parameters

Whether you are beginning a new simulation or modifying an existing one, it is a good idea to ensure that your simulation parameters are set exactly the way you want before you start processing. Once you Start a Simulation and begin processing, you cannot go back and change the settings without losing the simulation results that have been created.

Setting simulation parameters involves setting simulation-wide values as well as parameters specific to particles, geometries, materials, interactions, inputs, and more. To make the setup of multiple similar items quicker, you many also choose to duplicate already-setup items, or remove many similar items at one time.

SIMULATION PARAMETERS

You set simulation-wide parameters when you want to change the default values provided by Rocky to affect your whole simulation. Simulation-wide parameters are settings that can include:

- **Study** items, which include setting the simulation title and customer name.
- **Physics** items, which include setting how you want gravity applied; and setting simulation-wide models such as adhesion, rolling resistance, heat transfer, and coarse graining.
- **Modules** items, which include collision and particle statistics collection.
- **Domain Settings** items, which include defining the simulation coordinate limits and setting an optional periodic domain.
- **Solver** items, which include setting simulation time length, and certain data collection times.

Simulation-wide parameters are set by first selecting the Study, Physics, Modules, Domain Settings, and Solver sections in the Data panel and then editing the results in the Data Editors panel. These values can be set at any time before you begin processing your simulation. However, it is recommended that at least the Physics and Modules settings be made first as your selections there may affect other settings later on.

What would you like to do?

- Learn more Study Parameters (p. 415)
- Learn more Physics Parameters (p. 417)
- Learn more Modules Parameters (p. 431)
- Learn more SPH Parameters (p. 436)
- Learn more Domain Settings Parameters (p. 452)
- Learn more Solver Parameters (p. 464)

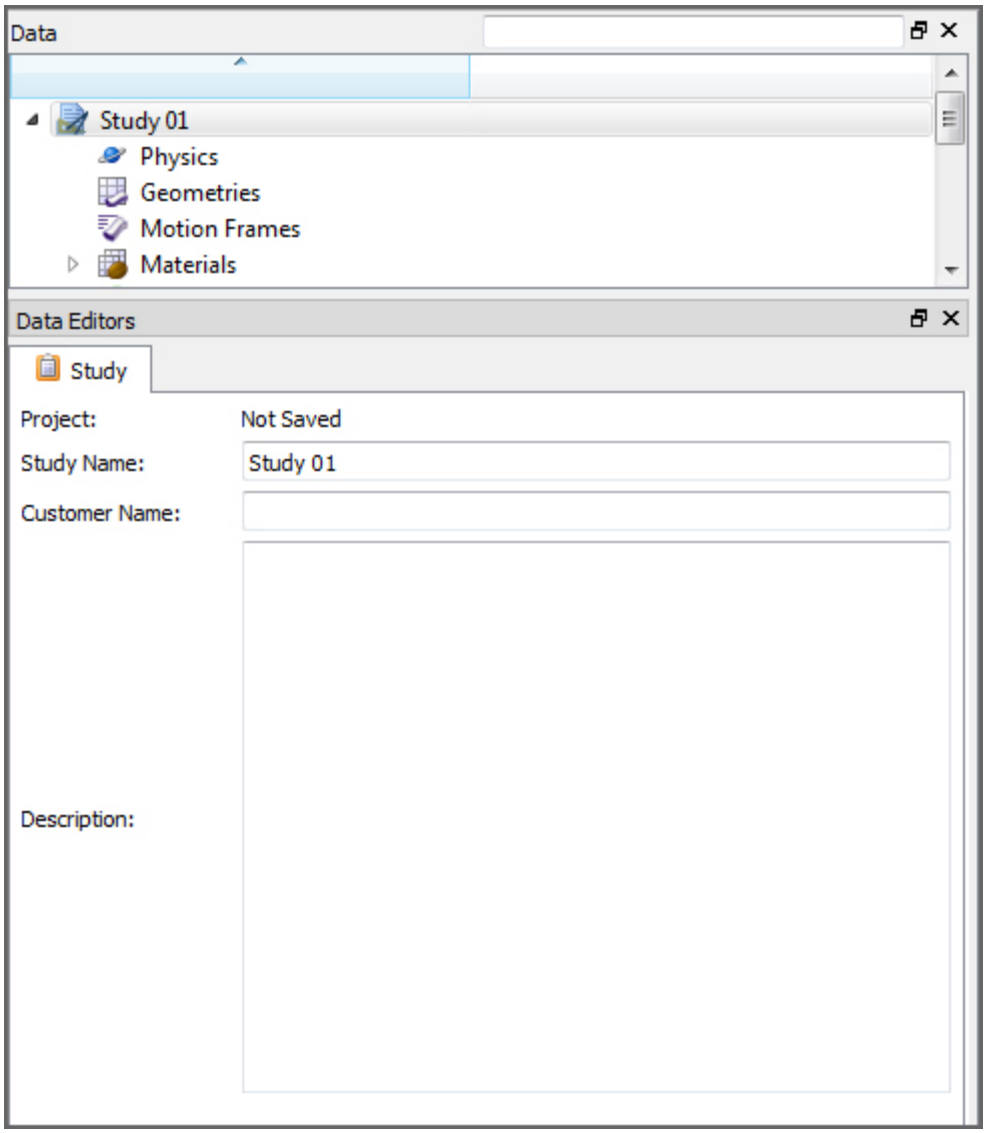
Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

3.2.1. Study Parameters

Note: Unlike most other setup parameters, you are able to change your Study parameters at any time-even during active processing. (See also I cannot change my setup parameters during processing (p. 1361).)

Use the figures and table below to help you understand the various Study parameters you can set for a simulation project.

Figure 3.1: Study parameters in the Data Editors panel (default, unsaved project)



Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

Figure 3.2: Study parameters in the Data Editors panel (saved project with information entered)

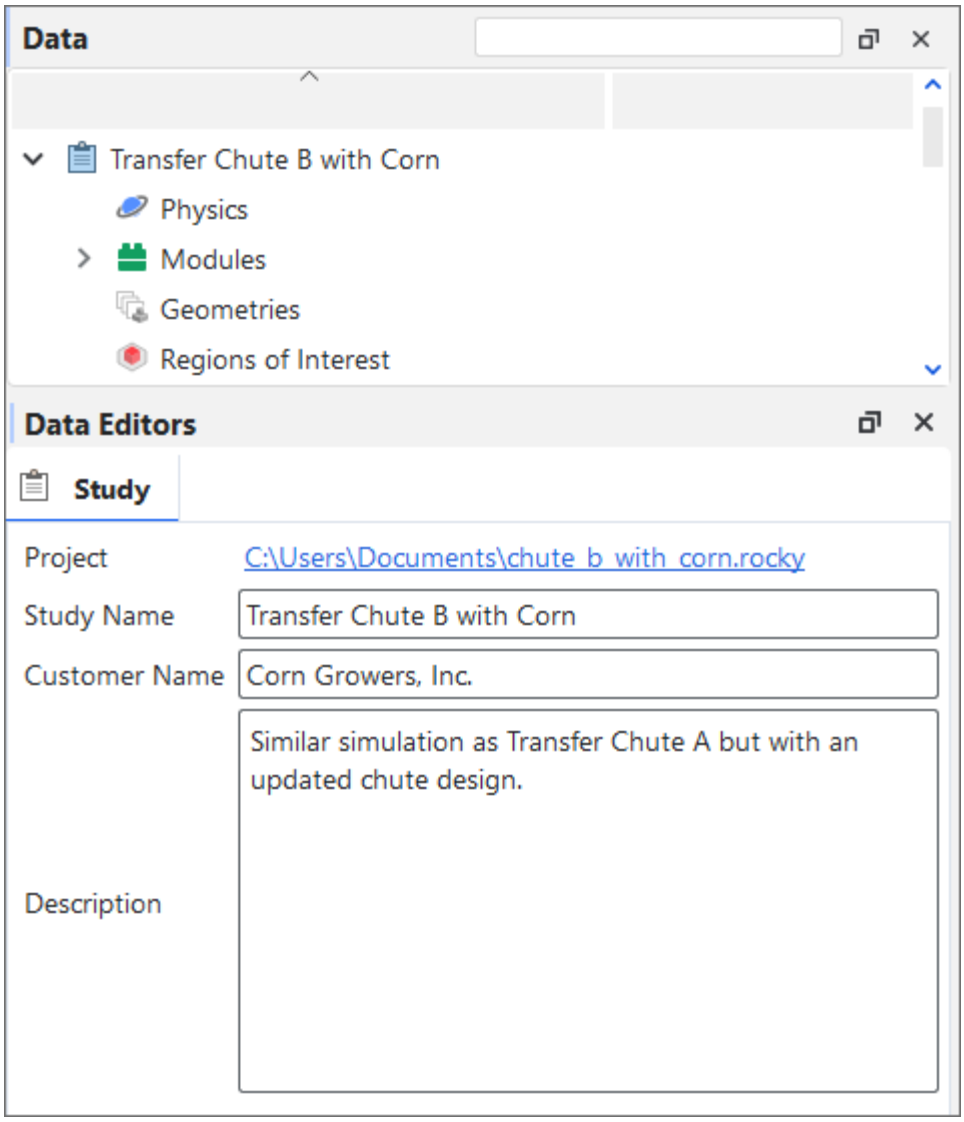


Table 1: Study parameter options

Setting	Description	Range
Project	When the project is saved, this lists the full path for the .rocky project file in hyperlink form (Figure 2). If the project has not yet been saved, "Not Saved" will be displayed (Figure 1).	Path automatically provided
Study Name	Name of the simulation you are working on. For example, "Transfer Chute B with corn." **Note:** The Study entity on the Data panel will be renamed with what you enter here (Figure 2).	No limit

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

Setting	Description	Range
Customer Name	Name of the customer for whom you are doing the simulation.	No limit
Description	Description of the simulation.	No limit

See Also:

- About Physics Parameters (p. 417)
- About Modules Parameters (p. 431)
- About Domain Settings Parameters (p. 452)
- About Solver Parameters (p. 464)
- Set Simulation Parameters (p. 414)

3.2.2. Physics Parameters

The Rocky Physics parameters include simulation-wide settings that affect how the components are calculated. These include settings affecting gravity; the rolling resistance and force models used for calculating momentum; and the separate settings that enable Thermal Modeling and Coarse-Graining.

MOMENTUM MODELS

The rolling resistance and force models used for calculating momentum have specific combinations that are unsupported in this version of Rocky. Refer to the compatibility table in Physics and Force Limitations (p. 55) for details.

To learn more about how these models are calculated, refer to the DEM Technical Manual. (From the Rocky **Help** menu, point to **Manuals** and then click **DEM Technical Manual**.)

THERMAL MODELING

To learn more about setting up and using Thermal in Rocky, refer to the Enable Thermal Modeling Calculations (p. 400) topic.

To learn more about how this model is calculated, refer to the DEM Technical Manual. (From the Rocky **Help** menu, point to **Manuals** and then click **DEM Technical Manual**.)

COARSE-GRAINING

To learn more about the Coarse-Graining used in Rocky, and see examples for its usage, refer to the following resources:

- DEM Technical Manual. (From the Rocky **Help** menu, point to **Manuals** and then click **DEM Technical Manual**.)
- CFD Coupling Technical Manual. (From the Rocky **Help** menu, point to **Manuals** and then click **CFD Coupling Technical Manual**.)
- Tutorial - Mixing Tee in the *Rocky Tutorial Guide*.

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

LIMITATIONS FOR COARSE-GRAINING

In this version of Rocky, Coarse-Graining and Particle Breakage are incompatible. This means if any *Particle sets* (p. 1393) within an Enable Coarse-Graining (previously named Coarse Grain Modeling (CGM)) simulation has breakage parameters enabled, the entire simulation-including any Particle sets without breakage-will be unable to be processed.

In addition, CGM works only with single-element particle shapes. This means that Coarse Grain Models are incompatible with Particle sets composed of multiple elements (also known as Meshed (p. 1392) particles), including flexible Fibers (p. 1391), flexible Shells (p. 1395), and flexible Solids (p. 1395).

In terms of momentum settings, CGM can be used only with these two Adhesive Forces: Constant and Linear. All other types of Adhesive Forces-including JKR-are incompatible with Coarse Grain Modeling in this version. Also note that the Velocity Dependent restitution model (an Experimental (Beta) Feature) is also incompatible with CGM.

In terms of CFD coupling, Coarse Graining is incompatible with **1-Way LBM**. While other types of coupling is allowed, only drag forces will be considered. This means that fluid forces such as torque, virtual mass, and lift are not compatible with Coarse Graining. (See also the **CFD Coupling Technical Manual**. (From the Rocky **Help** menu, point to **Manuals** and then click **CFD Coupling Technical Manual**.))

Also note that the Radl et al. Coarse Grain Model is currently incompatible with Multi GPU processing. (It is compatible with both Single GPU and CPU processing, however.)

(See also Physics and Force Limitations (p. 55).)

PHYSICS PARAMETER DEFINITIONS

Use the figures and table below to help you understand the various Physics parameters you can set for a simulation project.

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

Figure 3.3: Physics, Gravity parameters in the Data Editors panel

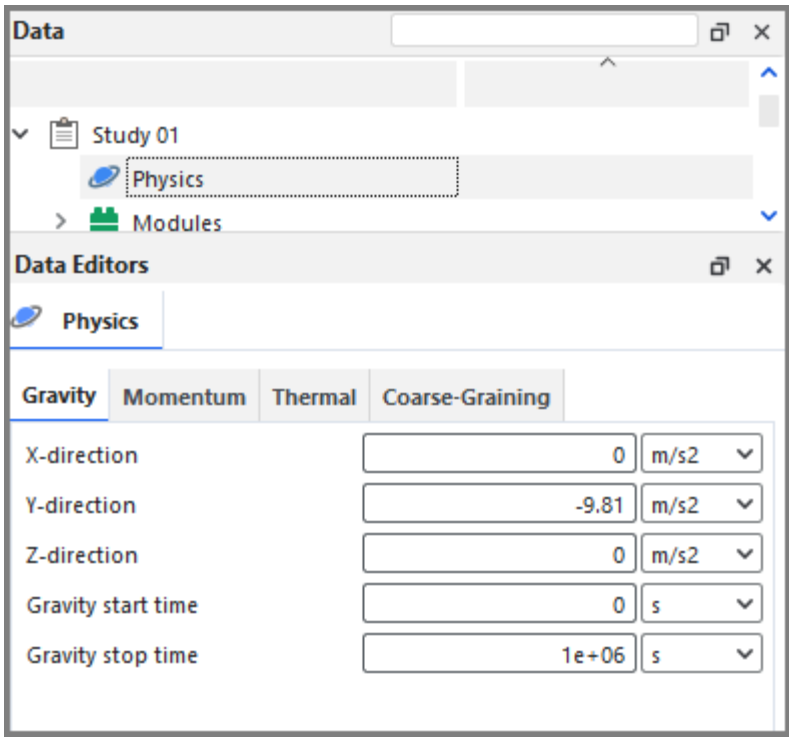
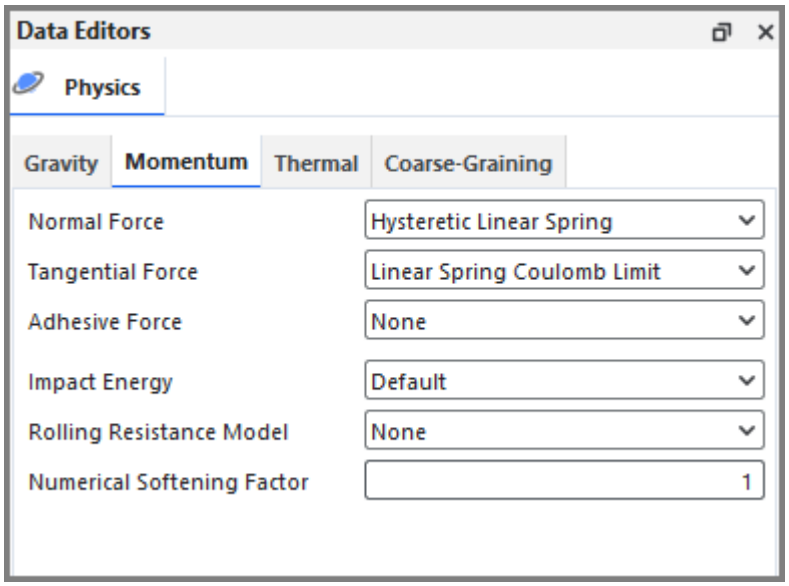


Figure 3.4: Physics, Momentum parameters in the Data Editors panel



Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

Figure 3.5: Physics, Thermal parameters in the Data Editors panel

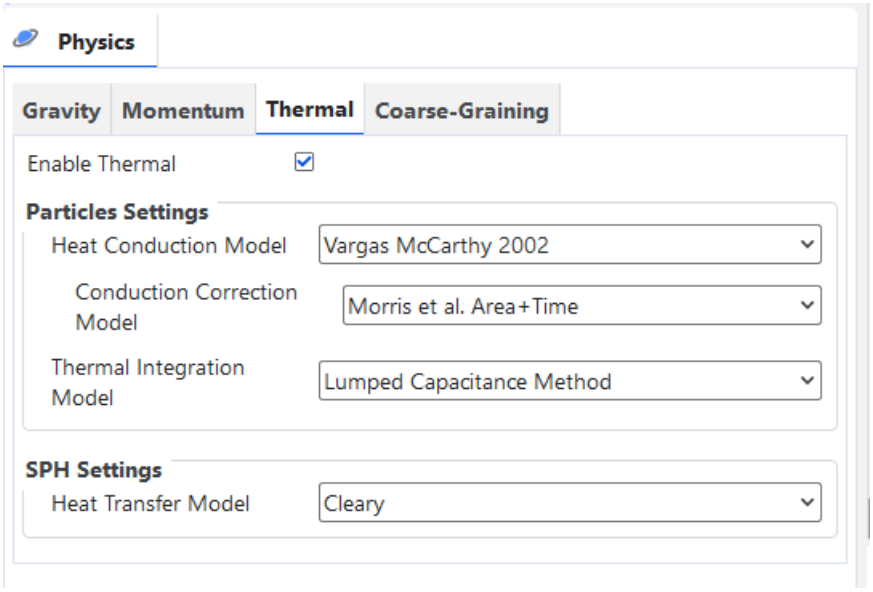


Figure 3.6: Physics, Coarse-Graining parameters in the Data Editors panel

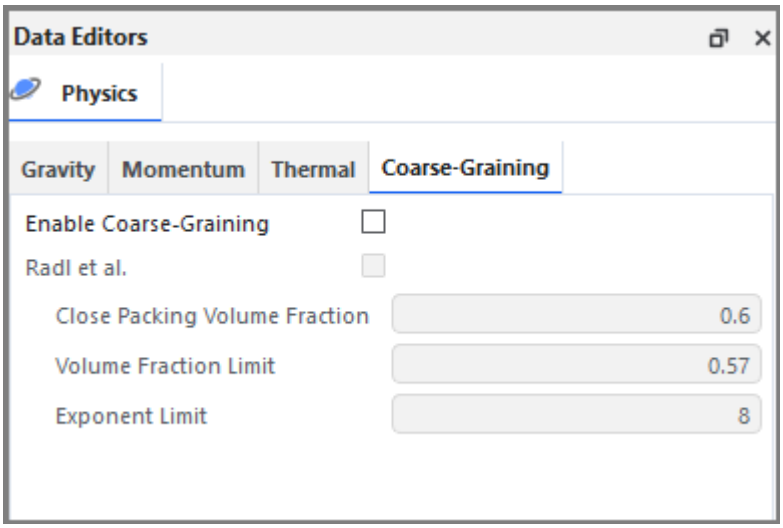


Table 1: Physics parameter options (all tabs)

Setting	Description	Range
Gravity		
X-direction	Used to change the direction that gravity affects particles and free boundaries, this is the amount of acceleration applied in the X direction during the simulation. **Tip:** When prescribing movements, it can be easier to align geometries with the	No limit

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

Setting	Description	Range
	global axes and then simulate gravity in the plane that represents downward forces. For example, if you had your equipment horizontally aligned with the X direction, you could then modify the Y and X components of gravity to simulate it as if it were inclined in the YX plane.	
Y-direction	Used to change the direction that gravity affects particles and free boundaries, this is the amount of acceleration applied in the Y direction during the simulation. **Note:** The default value is -9.81 m/s2, which accounts for the effect of gravity pointing in the downward Y direction only.	No limit
Z-direction	Used to change the direction that gravity affects particles and free boundaries, this is the amount of acceleration applied in the Z direction during the simulation.	No limit
Gravity Start Time	The duration you want to wait before gravity components are activated.	Positive values
Gravity Stop Time	The duration you want to wait before gravity components are deactivated.	Positive values
Momentum		
Normal Force	The type of model used to calculate the normal components of the *contact* (p. 1390) forces. The model you select here determines what **Adhesive Force** and **Tangential Force** options are available. For additional details about these models, see the Rocky DEM Technical Manual. (From the Rocky program **Help ** menu, point to **Manuals**, and then click **DEM Technical Manual**.)	Hysteretic Linear Spring; Linear Spring Dashpot; Hertzian Spring Dashpot Note: This setting is considered to be *exclusive* (p. 1391) so if you have one or more *external Modules* (p. 1391) that are able to override this model, you must have only one such Module enabled within your simulation project.

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

Setting	Description	Range
		Refer to the Module's documentation (if provided) for more information. (See also Rocky Simulation Entities that can be Affected by Modules (p. 223).)
Tangential Force	The type of model used to calculate the tangential components of the *contact* (p. 1390) forces. The models available are dependent upon the **Normal Force** you selected. Specifically: • **Linear Spring Coulomb Limit** works with all three Normal Force options. • **Coulomb Limit** works with all three Normal Force options. Note: **Coulomb Limit** does not work with the **Rolling Resistance Model** of **Type C: Linear Spring Rolling Limit**. • **Mindlin-Deresiewicz ** works only with the **Hertzian Spring Dashpot** Normal Force option. For additional details about these models, see the Rocky DEM Technical Manual. (From the Rocky program **Help ** menu, point to **Manuals**, and then click **DEM Technical Manual**.)	Linear Spring Coulomb Limit; Coulomb Limit; Mindlin-Deresiewicz Note: This setting is considered to be *exclusive* (p. 1391) so if you have one or more *external Modules* (p. 1391) that are able to override this model, you must have only one such Module enabled within your simulation project. Refer to the Module's documentation (if provided) for more information. (See also Rocky Simulation Entities that can be Affected by Modules (p. 223).)
Adhesive Force	Defines how adhesion between materials is calculated. Material-to- Material interactions are then defined in the Modify Material Interaction and Adhesion Values step. (See also About Modifying Materials Interactions and Adhesion Values (p. 597).) The models available are dependent upon the **Normal Force** you selected. Specifically: • **None** specifies that no adhesive force will be used in the simulation. As a result there will be no Adhesive properties to set for Material Interactions. • **Constant** works with all three Normal Force options.	None; Constant; Linear; JKR Note: This setting is considered to be *exclusive* (p. 1391) so if you have one or more *external Modules* (p. 1391) that are able to override this model, you must have only one such Module enabled within your simulation project. Refer to the Module's

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

Setting	Description	Range
	• **Linear** works with all three Normal Force options. • **JKR** works only with the **Hertzian Spring Dashpot** Normal Force option. Note: This model is incompatible with Enable Coarse-Graining. (See also the **Coarse-Graining** section below For additional details about these models, see the Rocky DEM Technical Manual. (From the Rocky program **Help ** menu, point to **Manuals**, and then click **DEM Technical Manual**.)	documentation (if provided) for more information. (See also Rocky Simulation Entities that can be Affected by Modules (p. 223).)
Restitution Model	When the **Experimental (Beta) Features** checkbox is enabled on the **Options | Preferences** dialog (see also About Setting Global Preferences (p. 227)), this enables you to select the type of model used to calculate restitution. **Note:** The Velocity Dependent model is incompatible with Enable Coarse-Graining. (See also the **Coarse-Graining** section below.)	Constant; Velocity Dependent
Impact Energy	The type of model used to calculate the impact energy of a *contact* (p. 1390). Unless you have enabled a custom *external Module* (p. 1391) that defines a different Impact Energy model, the only option listed will be **Default**. Impact energy is used in Rocky as the main input parameter for the built-in instantaneous breakage models. You will need to implement a custom impact energy calculation only if you intend to use a custom contact force model along with such breakage models. If in that case, you choose not to implement the impact energy calculation, Rocky will use a standard calculation based on the impact work, as defined in equation (4.10) of the DEM Technical Manual.	Default Note: This setting is considered to be *exclusive* (p. 1391) so if you have one or more *external Modules* (p. 1391) that are able to override this model, you must have only one such Module enabled within your simulation project. Refer to the Module's documentation (if provided) for more information. (See also Rocky Simulation Entities that can be Affected by Modules (p. 223).)
Rolling Resistance Model	Defines how rolling resistance for particles are calculated. After a model is selected, **Rolling**	None; Type A: Constant Moment;

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

Setting	Description	Range
	Resistance values are then set per *Particle sets* (p. 1393) on its **Movement** tab. (See also About Adding and Editing Particle Sets (p. 602)). Specifically: • **None**: No Rolling Resistance model will be used in the simulation. As a result, you will be unable to set a **Rolling Resistance** value for your *Particle sets* (p. 1393). • **Type A: Constant Moment**: Also known as a "Type 1" model, this is a simple model typically used by other DEM programs. This model should only be used when you want a high angle of repose without using adhesion values. • **Type C: Linear Spring Rolling Limit**: Also known as a "Type 3" model, in this model the rolling resistance moment initially increases linearly with the relative rotation between the contacting entities. When a limiting value proportional to the rolling resistance is reached, the rolling resistance moment becomes constant. This is the model recommended for most simulations that require reproducing rolling resistance effects. Note: The Type C model cannot be used with the **Coulomb Limit** model for **Tangential Force**. **Note:** Both types of Rolling Resistance Models are incompatible with *concave* (p. 1390) Custom Polyhedron particle shapes (see also Physics and Force Limitations (p. 55).) For more information on the calculations used to form these models, see the Rocky DEM Technical Manual. (From the Rocky program **Help ** menu, point to **Manuals**, and then click **DEM Technical Manual**.)	Type C: Linear Spring Rolling Limit
Numerical Softening Factor	Factor applied to the materials (includes both particles and boundaries) stiffnesses values that are used to calculate both time steps and contact forces. This enables you increase the size of your time steps, and speed up your processing time, without having to change the material properties themselves. This is especially useful in long or particularly complex cases, or for those with Thermal Modeling enabled.	Positive values Note: Typical values will range between 0.001 and 1.0

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

Setting	Description	Range
	Tips: • The smaller the value, the larger the simulation time step, which increases processing speed. • Values too small can lead to particle overlaps, simulation instability, and inaccurate results. See also About Modifying Material Compositions (p. 591) and and Enable Thermal Modeling Calculations (p. 400).	
Thermal		
Enable Thermal	Makes possible the simulation of conductive heat transfer between particles and other particles, and particles and boundaries. Enables convective heat transfer between particles and fluids when coupled with Ansys Fluent (see also Set or Modify Fluid and/or Air Flow Properties (p. 757)). See also Enable Thermal Modeling Calculations (p. 400).	Turns on or off
Particle Settings		
Heat Conduction Model	When **Enable Thermal** is selected, this is the type of model used to calculate heat conduction. Unless you have enabled a custom *external Module* (p. 1391) that defines a different Heat Conduction Model, only the default Vargas McCarthy 2002 model will be used.	Vargas McCarthy 2002 Note: This setting is considered to be *exclusive* (p. 1391) so if you have one or more *external Modules* (p. 1391) that are able to override this model, you must have only one such Module enabled within your simulation project. Refer to the Module's documentation (if provided) for more information. (See also Rocky Simulation Entities that can be Affected by Modules (p. 223).)
Conduction Correction Model	When **Thermal** is enabled, this allows you to select a strategy to counteract the adverse effects that can occur when you use the **Numerical Softening**	None; Morris et al. Area; Morris et al. Area+Time

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

Setting	Description	Range
	Factor to speed up a simulation. (See also the definition in the **Momentum** tab section above.) Specifically: • **None** will use no correction model. This might be best used when **Numerical Softening Factor** is kept as 1 (default). • **Morris et al. Area** is a correction model best used when **Numerical Softening Factor** is less than 1 and the thermal simulation is a static system with mostly resting particles. • **Morris et al. Area+Time** is a correction model best used when **Numerical Softening Factor** is less than 1 and the thermal simulation is a dynamic system with mostly non-resting particles. **Tip:** To learn more about these models, refer to the DEM Technical Manual. (From the Rocky **Help** menu, point to **Manuals** and then click **DEM Technical Manual**.)	
Thermal Integration Model	When **Enable Thermal** is selected, this is the type of model used to integrate heat conduction calculations. Unless you have enabled a custom *external Module* (p. 1391) that defines a different Thermal Integration Model, only the default **Lumped** Capacitance Method will be used, which ignores any conduction inside the particle and assumes only that the entire particle has a single temperature.	Lumped Note: This setting is considered to be *exclusive* (p. 1391) so if you have one or more *external Modules* (p. 1391) that are able to override this model, you must have only one such Module enabled within your simulation project. Refer to the Module's documentation (if provided) for more information. (See also Rocky Simulation Entities that can be Affected by Modules (p. 223).)
SPH Settings		

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

Setting	Description	Range
Heat transfer Model	When **Enable Thermal** is selected, this is the type of model used to calculate heat transfer for SPH elements. Unless you have enabled a custom external module that defines a different Heat transfer Model, only the default, Cleary model, will be used.	Cleary **Note:** This setting is considered to be ***exclusive*** so if you have one or more ***external Modules*** that are able to override this model, you must have only one such Module enabled within your simulation project. Refer to the Module's documentation (if provided) for more information.
Coarse-Graining		
Enable Coarse-Graining	Enables a coarse grain model (CGM) to be used in the simulation. **Tip:** Turning on Enable Coarse-Graining allows you to increase the original particle size and reproduce the system behavior using a reduced number of particles via use of the **CGM Scale**	Turns on or off

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

Setting	Description	Range
	Factor. (See also the *Size Tab* section of About Adding and Editing Particle Sets (p. 602).) Notes: In this version of Rocky, Enable Coarse-Graining is incompatible with: • JKR Adhesion Model (See also the **Adhesive Force** section above). • Velocity Dependent Restitution Model (See **Restitution Model** section above). • Particle Breakage (See also About Particle Breakage (p. 665)). • Particles composed of multiple elements- also known as *meshed* (p. 1392) or flexible particles (See also About Adding and Editing Particle Sets (p. 602)). • 1-Way LBM for simulating air flow (See also About Using the 1-Way LBM Method (p. 771)). • SPH (Smoothed-Particle Hydrodynamics).	
Radl et al.	When **Enable Coarse-Graining** is selected, this determines whether or not the Radl et al. model will be enabled, which helps describe additional physical processes on particle flows by determining the amount of energy dissipated based upon the velocities of the surrounding particles. Specifically: • When cleared, only the model from Bierwisch et al. (2009) will be used. • When enabled, both the model from Bierwisch et al. (2009) and the model from Radl et al. (2011) will be used. **Note:** In this version of Rocky, the Radl et al. model is incompatible with Multi GPU	Turns on or off

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

Setting	Description	Range
	processing. (See also About Starting a Simulation (p. 908).) **Tip:** To learn more about these models, refer to the DEM Technical Manual. (From the Rocky **Help** menu, point to **Manuals** and then click **DEM Technical Manual**.)	
Close Packing Volume Fraction	When **Radl et al.** is enabled, this defines the maximum volume fraction that particles are allowed to reach, which is one of three parameters that together determine what characteristics turn off the Radl et al. model and how quickly that shutoff happens. The shutoff function is especially important for cases with closely packed particles in constant contact as the Radl et al. model is not optimized for this scenario and applying it in those situations result in too much energy being removed from the system. **Tip:** This parameter is represented by $\alpha_{p,cp}$ in equation 2.101 of the Rocky DEM Technical Manual. **Note:** This text field does not support parametric variables. (See also I cannot enter an input variable or mathematical function into a text field (p. 1334).)	0-1
Volume Fraction Limit	When **Radl et al.** is enabled, this defines the particle volume fraction value above which the shutoff function will not affect the particles' relaxation time. This parameter is one of three that together determines what characteristics turn off the Radl et al. model and how quickly that shutoff happens.	0-1

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

Setting	Description	Range
	The default value of 0.57 represents 95% of the default **Close Packing Volume Fraction** of 0.6. **Tip:** This parameter is represented by $\alpha_{p,off}$ in equation 2.101 of the Rocky DEM Technical Manual.	
Exponent Limit	When **Radl et al.** is enabled, this defines the exponent value (default is 8) used in the shutoff function. The exponent is used to control how sensitive the shutoff function is regarding volume fraction variations, and is one of three parameters that together determines what characteristics turn off the Radl et al. model and how quickly that shutoff happens. **Tip:** Refer to equation 2.101 in the Rocky DEM Technical Manual for details.	0-100
Search Distance Multiplier	When the **Advanced Features** checkbox is enabled on the **Options** | **Preferences** dialog (see also About Setting Global Preferences (p. 227)), and the **Radl et al.** checkbox is also enabled, this value will be multiplied by the biggest particle size to achieve the estimation radius used by the Radl et al. model to determine which particles are close enough to be used for energy dissipation calculations. Specifically: • Values larger than 3 achieve a larger radius and thereby include more particles when computing the average neighbor velocity. However, this may also increase the computational cost. • Values less than 3 but greater than 0 (zero) reduce the radius and thereby include only the nearest particles when computing neighbor velocity.	Positive values **Note:** This value must be greater than **Update Distance Multiplier**

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

Setting	Description	Range
Updated Distance Multiplier	When the **Advanced Features** checkbox is enabled on the **Options** | **Preferences** dialog (see also About Setting Global Preferences (p. 227)), and the **Radl et al.** checkbox is also enabled, this value will be multiplied by the biggest particle size to define the distance a particle has to move before Rocky will update the neighbor list used by the Radl et al. model to compute the average neighbor velocity. Specifically: • Values greater than 1 achieve a lower frequency at which the neighbor list is updated. • Values less than 1 achieve a higher frequency at which the neighbor list is updated, which can increase the computational cost.	Positive values **Note:** This value must be less than **Search Distance Multiplier**

What would you like to do?

- Learn more about Physics and Force Limitations (p. 55)

See Also:

- About Study Parameters (p. 415)
- About Modules Parameters (p. 431)
- About Domain Settings Parameters (p. 452)
- About Solver Parameters (p. 464)
- Set Simulation Parameters (p. 414)

3.2.3. Modules Parameters

When you enable a Module (p. 1392) in Rocky, you are choosing to add in custom, discrete features and/or functionality within your project. Depending upon the type of Module you enable and what it does, there may be additional settings that affect other areas of your Rocky project setup.

Use this topic to understand more about Module parameters and how setting them affect your project.

Tip:

To learn more about the different types of Rocky Modules and where you obtain them, refer to the topic Understand Rocky Modules (p. 220).

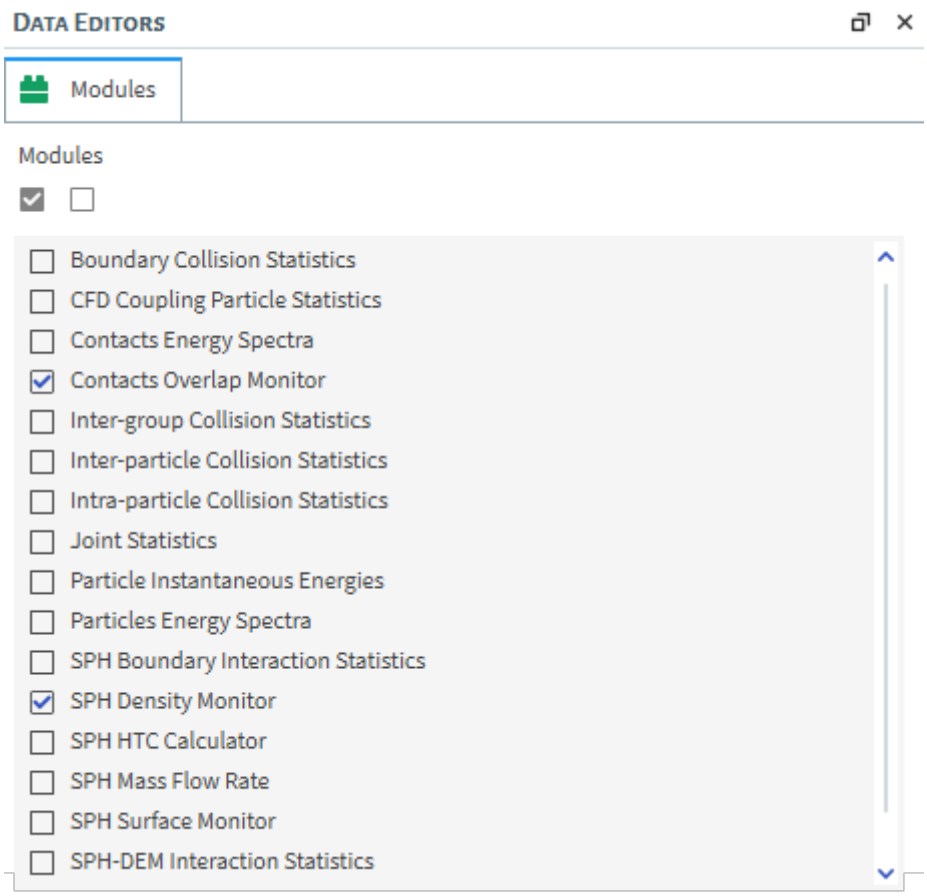
ENABLING MODULES

Because the default state for most Modules is disabled, it becomes very important for you to ensure that you have enabled the Modules and options you want prior to setting up the rest your simulation.

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

This is done by selecting the main **Modules** entity in the **Data** panel and then enabling the checkbox for the module you want to use in the **Data Editors** panel (Figure 1).

Figure 3.7: Modules available by default in Rocky



Important:

Important: Only embedded Modules (p. 1390) and external Modules (p. 1391) that you have already installed will appear in on the list. (See also Install an External Module (p. 223).)

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

In addition, because it is possible that turning on a Module will add or change options presented in other parts of the Rocky setup, it is recommended that turning on the Modules and options you want is one of the very first steps you take when setting up your project.

Tip:

If you have similar projects that you set up regularly, using a script to record and then play back your Module configuration steps can save you from having to repeat these steps on future projects. (See also About Creating and Using PrePost Scripts (p. 296).)

Note:

Enabling more than one Module at once can cause some parameters to be shared across Modules. (See also Recognize Shared Parameters with Asterisks (*) (p. 124).)

SETTING MODULE PARAMETERS

Once enabled, many types of Modules have parameters that you can define. Depending upon the type of Module and what it does, the settings can be in any of the following combinations.

Module Enables No Additional Settings

For these types of Modules, once you enable the Module itself, the functionality will be applied without any additional actions from you.

Tip:

You will know that a Module has no additional settings when you select it from the **Data** panel, and find no settings on the Module's **Data Editors** panel, and its **Info** tab says "Module does not affect any simulation entity."

An example of this kind of Module is the **Particle Instantaneous Energies** Module. (See also About the Particle Instantaneous Energies Module (p. 885).)

Module Enables Additional Settings

For these types of Modules, once you enable the Module itself, there are one or more additional settings in one or both of the following locations:

On the Module's Data Editors panel

For these types of Modules, there are one or more additional settings to be made on the **Data Editors** panel for the Module.

An example of this kind of Module is the **Boundary Collision Statistics** Module. (See also About the Boundary Collision Statistics Module (p. 868).)

Elsewhere in the Rocky setup

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

For these types of Modules, there are one or more additional settings to be made in (or have otherwise been affected by) other locations of the Rocky UI. These may require additional setup steps for the Module in those other locations.

Tip:

You will know that a Module has other settings in (or is otherwise affecting) other parts of the Rocky setup when you select it from the **Data** panel and then from the **Data Editors** panel, see that its **Info** tab has information next to the **Affected Simulation Entities** label. You can then use that information to discover where else in the Rocky setup Module-specific settings might need to be applied.

An example of this kind of Module is the **Liquid Bridge Model** Module, which has Module-specific settings in three other areas of the Rocky setup.

Note:

In this version of Rocky, the **Liquid Bridge Model** Module is provided as an *external* (p. 1391) module. Refer to the Install an External Module (p. 223) topic for details.

MODULE PARAMETERS LOCATIONS

Use this section to learn more about where and when you might define parameters for your Modules.

MAIN TAB FOR MODULES

Once enabled, some types of Modules will have additional settings on the Module's main tab in the **Data Editors** panel. To learn if there are additional, Module-specific settings that affect other parts of your Rocky project setup, you can view the information on the **Info** tab.

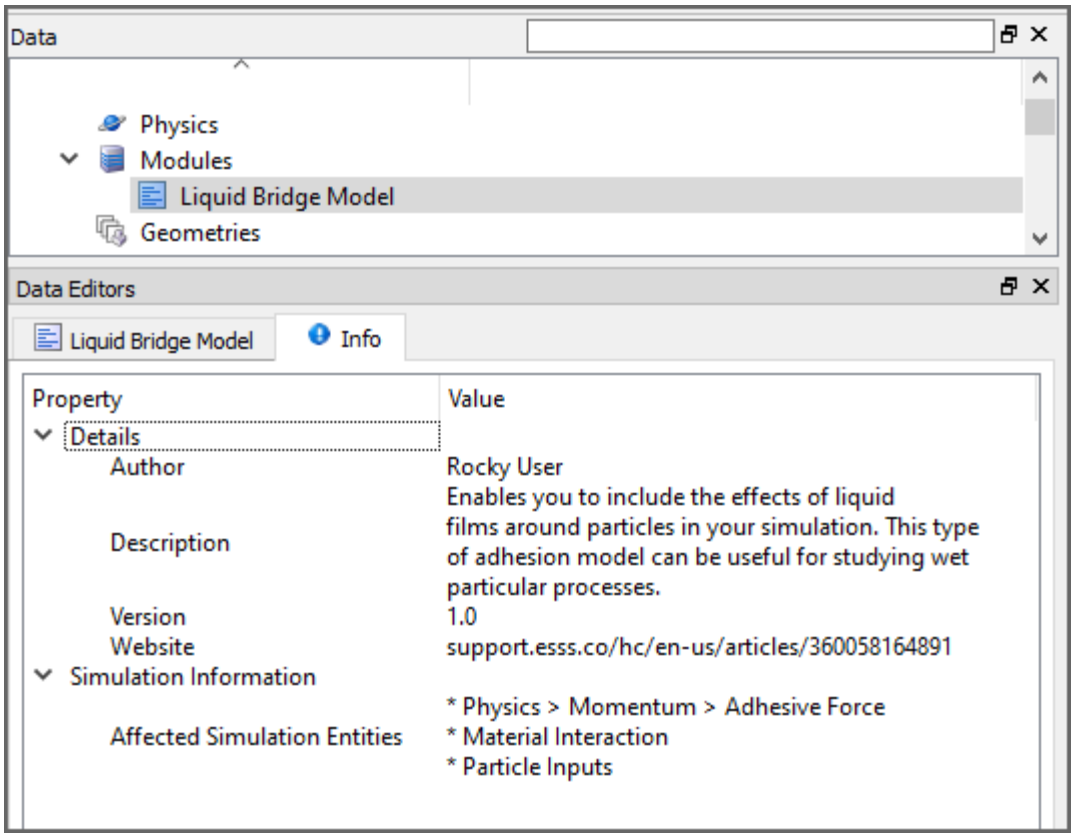
INFO TAB FOR MODULES

The Info tab on individual Modules (see also About the Info Tabs (p. 83)) describe the Author and Version details for the Module, and lists what other entities in the Rocky UI are affected by enabling that particular Module.

For example, the Info tab for the Liquid Bridge Model Module (Figure 2) describes three places in the Rocky UI affected by that module.

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

Figure 3.8: Info tab for the Liquid Bridge Model Module



This information can help you verify that the parameters in those locations are set correctly for the feature enabled by the module.

Note:

In this version of Rocky, the **Liquid Bridge Model** Module is provided as an *external* (p. 1391) module. Refer to the Install an External Module (p. 223) topic for details.

MODULE-SPECIFIC SETTINGS IN OTHER ROCKY UI LOCATIONS

Once enabled, some Modules will cause other areas of the Rocky UI to have additional settings specific to that Module, or will cause other changes, such as model overrides. These kinds of changes and additional settings are unique to each Module, so the best way to determine what other settings might be required for your Module is to view the **Affected Simulation Entities** information on the **Info** tab. (See the "About the Info Tab for Modules" section above.)

Once you understand what other parts of the UI are affected, you can then be sure to verify the Module-specific settings in those areas when setting up the rest of your project.

Tip:

To learn more about what specific Rocky UI settings and options Modules can affect, refer to the Rocky Simulation Entities that can be Affected by Modules (p. 223) topic.

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

What would you like to do?

- Learn more About Rocky Modules (p. 221)
- Learn more About the Boundary Collision Statistics Module (p. 868)
- Learn more About the Contacts Energy Spectra Module (p. 870)
- Learn more About the Contacts Overlap Monitor Module (p. 872)
- Learn more About the CFD Coupling Particle Statistics Module (p. 873)
- Learn more About the Inter-group Collision Statistics Module (p. 875)
- Learn more About the Inter-particle Collision Statistics Module (p. 878)
- Learn more About the Intra-particle Collision Statistics Module (p. 880)
- Learn more About the Particle Instantaneous Energies Module (p. 885)
- Learn more About Joint Statistics Module (p. 882)
- Learn more About the Particles Energy Spectra Module (p. 885)
- Install an External Module (p. 223)

See Also:

- About Study Parameters (p. 415)
- About Physics Parameters (p. 417)
- About Domain Settings Parameters (p. 452)
- About Solver Parameters (p. 464)

3.2.4. SPH Parameters

The SPH parameters available define how the simulation of fluid flow is going to be calculated inside Rocky.

Rocky uses an SPH (Smoothed-Particle Hydrodynamics) technique to perform fluid flow simulations. To ensure that the simulation is a good representation of the physics involved, you have control of several parameters that set how the SPH method is going to work in a Rocky simulation. Below you can learn more about these parameters.

Note:

Simulation files from previous versions of Rocky that included SPH as a module are **not** compatible with new versions from 23R1 onwards.

To learn more about how this model is calculated, refer to the SPH Technical Manual. (From the Rocky **Help** menu, point to **Manuals** and then click **SPH Technical Manual**).

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

ABOUT THERMAL MODELING

Rocky includes an option for thermal modeling in an SPH simulation. To learn more about setting up and using SPH Thermal model in Rocky, refer to the Enable Thermal Modeling Calculations (p. 400) topic.

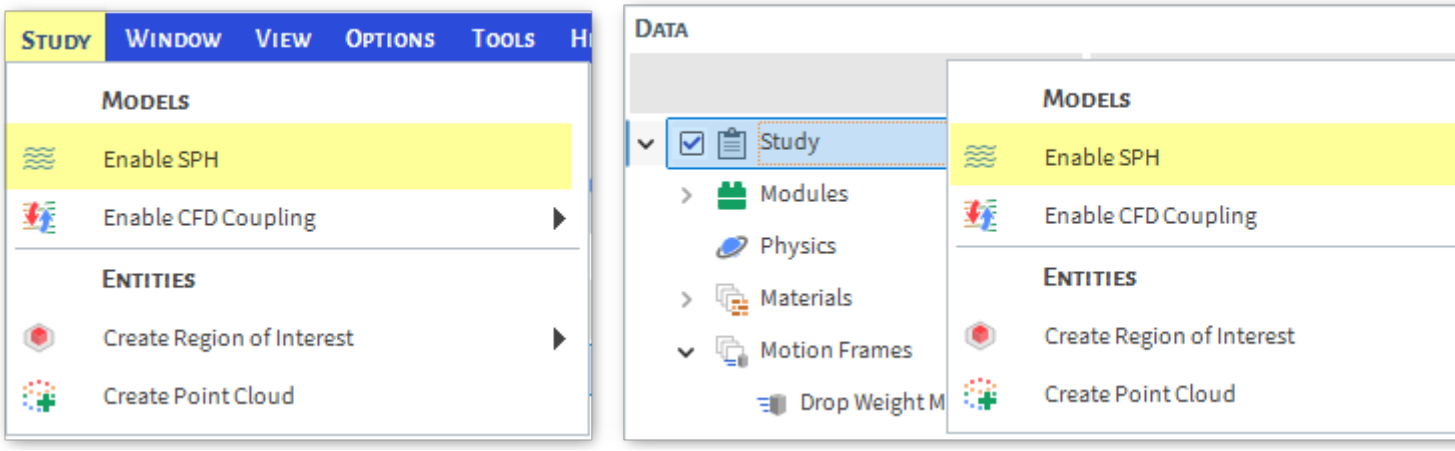
ABOUT EULERIAN SOLUTION

When setting up an SPH simulation, you have the option of using the Eulerian Solution feature. By default it is enabled. This option allows you to obtain interpolated results for SPH properties (see more About Properties (p. 933)). This option may result in more use of computational resources. If you want to disable it, you can do so by unchecking the checkbox **Eulerian Solution** under the **SPH** item on the data panel). To learn more about setting up and using Eulerian Solution in Rocky, refer to the About SPH Eulerian Solution (p. 400) topic.

SPH PARAMETERS AVAILABLE IN ANSYS ROCKY SOFTWARE

Important:

You can enable/disable SPH for your simulation either from **Menu | Study | Models** or by right-clicking the Study entity in the **Data panel** and selecting **Enable SPH**.



The SPH parameters in Rocky can be edited from the **SPH** entity on the data panel. They are divided into two categories: the **SPH Model Parameters** with the basic setup needed to perform an SPH simulation, and an **Advanced** tab, with parameters that allow a fine-tuning of the simulation.

SPH PARAMETERS DEFINITIONS

Use the figure and table below to help you understand the various SPH parameters you can set for a simulation project in the **SPH Model Parameters** tab.

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

Figure 3.9: SPH Model Parameters

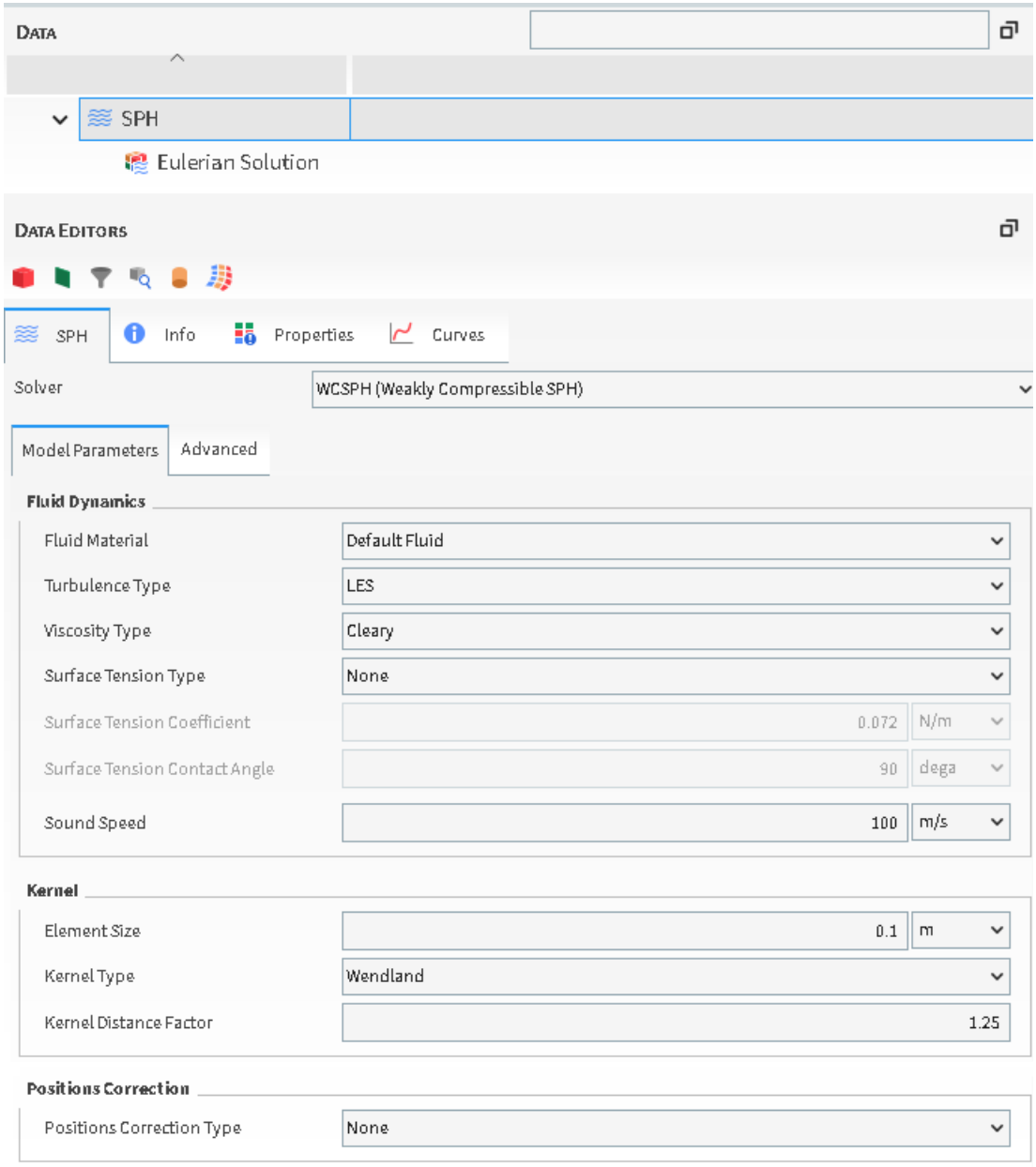


Table 1: SPH Solver Settings

Setting	Description
WCSPH (Weakly Compresible SPH)	The Weakly Compressible SPH solver is suitable for simulating incompressible fluids, by using an artificial equation of state to compute the relation between pressure and density.
ISPH (Implicit Incompressible SPH) BETA	The Implicit Incompressible SPH solver is also suitable for simulating incompressible fluids. In this formulation, the fluid incompressibility constraint is enforced by solving a system of linear equations at each simulation time, to compute pressure values that lead to element velocities and displacements that satisfy the constraint. This treatment allows to solve the flow equations

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

Setting	Description
	with larger time steps than the ones needed for the Weakly Compressible SPH (WCSPH) formulation. **Note:** The ISPH can only be used if the **Experimental (Beta) Features** checkbox is enabled on the **Options | Preferences | Additional Features** dialog. **Important:** Note that beta features have not been fully tested and validated. Ansys, Inc. makes no commitment to resolve defects reported against these prototype features. However, your feedback will help us improve the overall quality of the product. We will not guarantee that the projects using this beta feature will run successfully when the feature is finally released so you may, therefore, need to modify the projects.
DFSPH (Divergence-Free SPH) BETA	The Divergence Free SPH solver is also suitable for simulating incompressible fluids. This solver can be considered an extension of the IISPH formulation, because both enforce a constant density value through the solution of a Poisson pressure equation. However, DFSPH introduces an additional step in which the divergence-free condition is enforced on the velocity field. **Note:** The DFSPH can only be used if the **Experimental (Beta) Features** checkbox is enabled on the **Options | Preferences | Additional Features** dialog. **Important:** Note that beta features have not been fully tested and validated. Ansys, Inc. makes no commitment to resolve defects reported against these prototype features. However, your feedback will help us improve the overall quality of the product. We will not guarantee that the projects using this beta feature will run successfully when the feature is finally released so you may, therefore, need to modify the projects.

Note:

Simulations with parameters defined in ISPH as a module must be set again, as they will **not** be compatible with ISPH integrated in the Rocky UI from release 2024 R2 onwards.

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

Table 2: SPH Model Parameters Tab

Setting	Description	Range
Fluid Dynamics		
Fluid Material	Allows you to choose the fluid material that will be used in the simulation. (see also About Modifying Fluid Material Compositions (p. 595)).	Automatically Determined
Turbulence Type	Defines the turbulence modeling approach. • **Laminar**: indicated for low Reynolds number flows. • **LES**: turbulence model for turbulent flows based on the Large Eddy Simulation concept. For additional details about these models, see the SPH Technical Manual. (From the Rocky program **Help** menu, point to **Manuals**, and then click **SPH Technical Manual**.)	Laminar; LES
Viscosity Type	Defines the viscosity model used in the calculation of SPH element acceleration due viscous forces flow. • **Cleary**: indicated for fast turbulent flows. • **Morris**: indicated for slow laminar flows. For additional details about these models, see the SPH Technical Manual. (From the Rocky program **Help** menu, point to **Manuals**, and then click **SPH Technical Manual**.) **Important:** It is also important to mention that the Viscosity Type can be overruled by the SPH Turbulent Viscosity Limiter module, which defines a ratio between the set viscosity from the UI and the maximum viscosity that can be achieved. More information about how modules affect simulations can be found at Rocky Simulation Entities that can be Affected by Modules (p. 223). Also, information about this module can be found in its respective manual.	Cleary; Morris
Surface Tension Type	Defines the surface tension model used to model free surfaces. • **CSF**: Continuum Surface Force model, the most common model used in SPH simulations • **CSS**: Continuum Surface Stress model, alternative method that does not require knowledge of the surface curvature.	None; CSF; CSS; Pairwise Potential

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

Setting	Description	Range
	• **Pairwise Potential**: Enables simulations that detail the existence and shape of surface tension, along with its effects on fluid-fluid and fluid-solid interactions. If **CSF**, **CSS** or **Pairwise Potential** model are enabled, a new parameter will be required: • **Surface Tension Coefficient** (range: positive values) If **CSF** or **Pairwise Potential** model are enabled, a new parameter will be required: • **Surface Tension Boundary Angle** (range: positive values) If **Pairwise Potential** model is enabled, a new parameter will be required: • **Capillary Friction Coefficient** (range: numerical values between 0 and 1) **Important:** When the Pairwise Surface Tension type is selected, the **Surface Tension Boundary Angle** and the **Capillary Friction Coefficient** will be available for the specified geometry, located in the **Wall | SPH** tab (See Imported wall, SPH parameters in the Data Editors panel (p. 500)).	
Sound Speed / Maximum Expected Velocity	For WCSPH, the Sound Speed, which is about 10x the maximum expected velocity for the simulation must be set here. For IISPH and DFSPH, the value is the Maximum Expected Velocity for the simulation.	Positive Numbers [m/s]
Kernel		
Element Size	Defines the size of an SPH element. For coupled DEM-SPH simulations, an element size at least 3 times smaller than the smallest particle size in the simulation is necessary.	Positive Values
Kernel Type	Defines the type of kernel function used for SPH calculations • **Cubic**: Uses the Cubic Spline Kernel function as the interpolation scheme to compute a field variable at any point in a domain. • **Quintic**: Uses the Quintic Spline Kernel function as the interpolation scheme to compute a field variable at any point in a domain.	Cubic; Quintic; Wendland

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

Setting	Description	Range
	• **Wendland**: Uses the Wendland Spline Kernel function as the interpolation scheme to compute a field variable at any point in a domain. For additional details about these models, see the SPH Technical Manual. (From the Rocky program **Help** menu, point to **Manuals**, and then click **SPH Technical Manual**.)	
Kernel Distance Factor	Defines the kernel distance factor, which affects the size of the kernel support.	Positive Values
Positions Correction		
Positions Correction Type	Defines the formulation applied to prevent SPH elements from clumping together and distribute them more evenly in space. • **None**: No position corrections will be applied to the SPH elements • **XSPH**: The XSPH approach will be used to correct the position of the SPH elements • **Shift**: The Shifting approach *(Available only for WCSPH)* will be used to correct the position of the SPH elements For additional details about these models, see the SPH Technical Manual. (From the Rocky program **Help** menu, point to **Manuals**, and then click **SPH Technical Manual**.)	None; XSPH; Shift

SPH ADVANCED PARAMETER

Use the figure and table below to help you understand the various SPH parameters you can set for a simulation project in the **Advanced** tab.

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

Figure 3.10: SPH Model Advanced Parameters

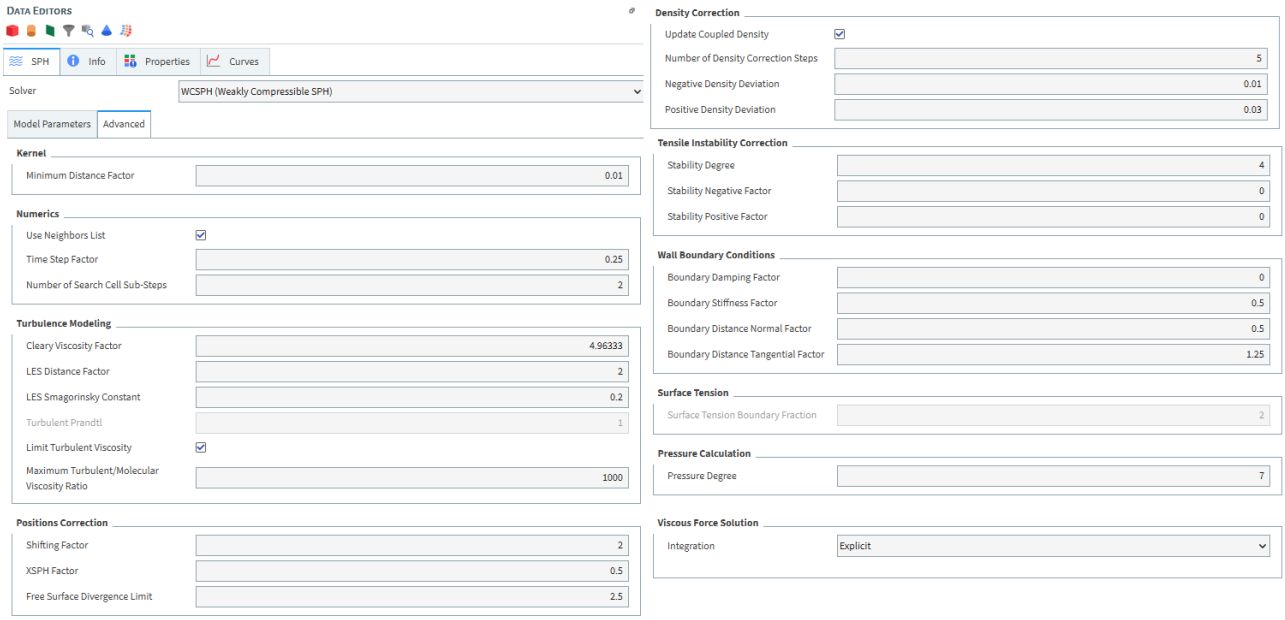


Table 3: SPH Advanced Parameters Tab

Setting	Description	Range
Kernel		
Minimum Distance Factor	Defines the factor used to calculate the minimum distance between SPH elements in order to avoid singularities.	Positive Values
Numerics		
User Neighbors List	Allows Rocky to use a neighbors list which is stored in memory. This will speed up the simulation, but if the number of elements in the list exceeds the system memory available, an allocation error may occur.	On or Off
Time Step Factor	Defines the coefficient used to compute the SPH solver time step.	Positive Values
Number of Search Cell Sub-Steps	Defines the coefficient that is used to define the search cell size.	Positive Values
Turbulence Modelling		
Cleary Viscosity Factor	Defines the coefficient used in the Cleary formulation of the acceleration due to viscous forces.	Positive Values
LES Distance Factor	Defines the characteristic length used to compute the turbulent viscosity term.	Positive Values
LES Smagorinsky Constant	Defines the Smagorinsky constant used to compute the turbulent viscosity term.	Positive Values
Limit Turbulent Viscosity	The Limit Turbulent Viscosity is used for clipping the viscosity at SPH simulations for turbulent	On or Off

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

Setting	Description	Range
	flows. This features aims at preventing the SPH turbulent viscosity from becoming too large and,therefore, possibly destabilizing the simulation.	
Maximum Turbulent/Molecular Viscosity Ratio	This parameter is used for the viscosity clipping, being the maximum viscosity ratio allowed for the current simulation.	Positive Values
Positions Correction		
Shifting Factor	Defines the shifting factor for the Shift approach.	Positive Values
XSPH Factor	Defines the XSPH factor used for the XSPH position correction model.	Positive Values
Free Surface Divergence Limit	Defines the concentration gradient reference used to avoid SPH elements placed near the free surface to be shifted in the normal direction for the Shifting particle correction algorithm.	Positive Values
Density Correction	Only for WCSPH	
Update Coupled Density	Defines if the density of the fluid will be updated with the calculations.	On or Off
Number of Density Correction Steps	Defines the frequency for the density correction calculation. **Note:** To turn off SPH Density Correction for SPH-only simulations, change the Number of Density Correction Steps to 0.	Positive Values
Negative Density Deviation	Defines the maximum negative pressure deviation allowed from the initial density value for an SPH element. **Note:** Another way to turn off SPH Density Correction for SPH-only simulations is to set Negative an Positive Density Deviation to 0.	Positive Values

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

Setting	Description	Range
Positive Density Deviation	Defines the maximum positive pressure deviation allowed from the initial density value for an SPH element. **Note:** Another way to turn off SPH Density Correction for SPH-only simulations is to set Negative and Positive Density Deviation to 0.	Positive Values
Tensile Instability Correction	Only for WCSPH	
Stability Degree	Defines the degree of the tensile instability correction.	Positive Values
Stability Negative Factor	Defines the coefficient that multiplies the pressure, if negative, for computing the tensile instability correction term. Default value is zero as the tensile instability correction was found unnecessary for most simulations with the current formulation.	Positive Values
Stability Positive Factor	Defines the coefficient that multiplies the pressure, if positive, for computing the tensile instability correction term. Default value is zero as the tensile instability correction was found unnecessary for most simulations with the current formulation.	Positive Values
Wall Boundary Conditions		
Boundary Damping Factor	Defines the viscous damping coefficient used in the calculation of the normal force acting on the SPH element due to the wall interaction.	Positive Values
Boundary Stiffness Factor	Defines the elastic coefficient used in the calculation of the normal force acting on the SPH element due to the wall interaction.	Positive Values
Boundary Distance Normal Factor	Defines a multiplier used to evaluate the interaction distance between SPH elements and boundaries for the normal force.	Positive Values
Boundary Distance Tangential Factor	Defines a multiplier used to evaluate the interaction distance between SPH elements and boundaries for the tangential force.	Positive Values
Surface Tension		
Surface Tension Boundary Fraction	Defines a factor used to calculate the surface tension.	Positive Values
Pressure Calculation	Only for WCSPH	

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

Setting	Description	Range
Pressure Degree	Coefficient of the equation of state used to compute the relation between pressure and density	Positive Values
Pressure Solution	Only for IISPH and DFSPH	
Density Relative Error Tolerance	Defines the tolerance for the relative error in the density calculation.	Positive Values
Maximum Number of Iterations	The maximum number of iterations for convergence of the pressure.	Positive Values
Pressure Under-Relaxation Factor	Defines a factor introduced to ensure stability of the process, by reducing the amount of changes between iterations.	Positive Values
Negative Pressure Factor	Defines the factor that multiplies negative pressures during the iterative solution process.	Positive Values
Relaxed Incompressibility Constraint	Only for IISPH **Note:** The **Relaxed Incompressibility Constraint** can only be used if the Experimental (Beta) Features checkbox is enabled on the Options | Preferences | Additional Features dialog.	On or Off
Incompressibility Relaxation Factor	Defines the factor that multiples the density invariance condition (constant density condition).	Positive Values
Viscous Force Solution		
Integration	Defines the method for viscous force integration.	Explicit or Implicit
Relative Error Tolerance	(Only for implicit) Defines the tolerance for the relative error in the viscosity calculation.	Positive Values
Maximum Number of Iterations	(Only for implicit) The maximum number of iterations for convergence of the viscosity.	Positive Values
Under-Relaxation Factor	(Only for implicit) Defines a factor introduced to ensure stability of the process, by reducing the amount of changes between iterations.	Positive Values

ABOUT SPH AND ROI

The current SPH feature also has a support for Region of Interest (ROI). This means that it is capable of freezing and disabling SPH-SPH interactions outside a given region. With this feature, there can be a simulation speed increase.

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

What would you like to do?

- Learn more about Physics and Force Limitations (p. 55)

See Also:

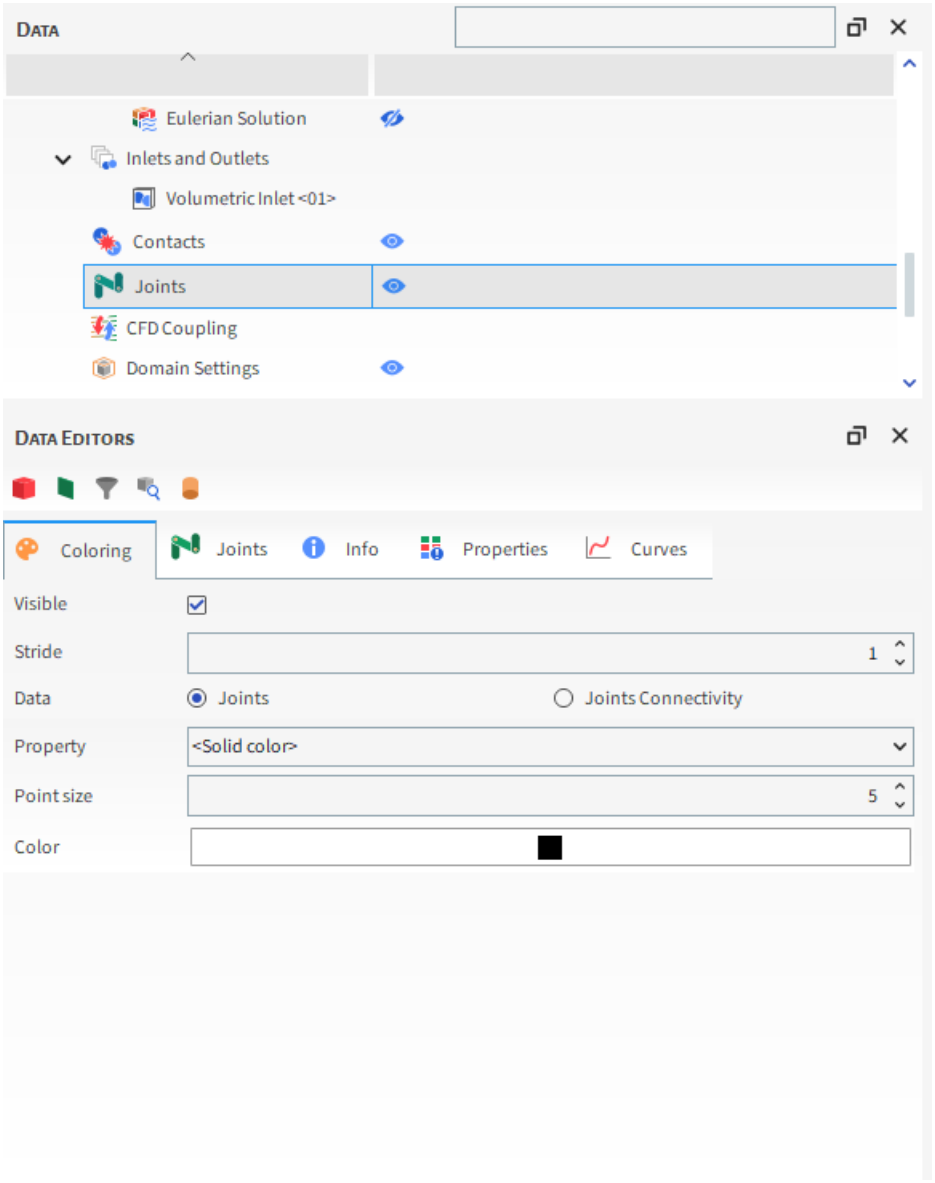
- About Study Parameters (p. 415)
- About Modules Parameters (p. 431)
- About Domain Settings Parameters (p. 452)
- About Solver Parameters (p. 464)
- Set Simulation Parameters (p. 414)

3.2.5. Joints Parameters

Use the figure below to help you understand the Joints parameters you can set for a simulation project.

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

Figure 3.11: Joints parameters in the Data Editors panel (Coloring - Joints)



Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

Figure 3.12: Joints parameters in the Data Editors panel (Coloring - Joints Connectivity)

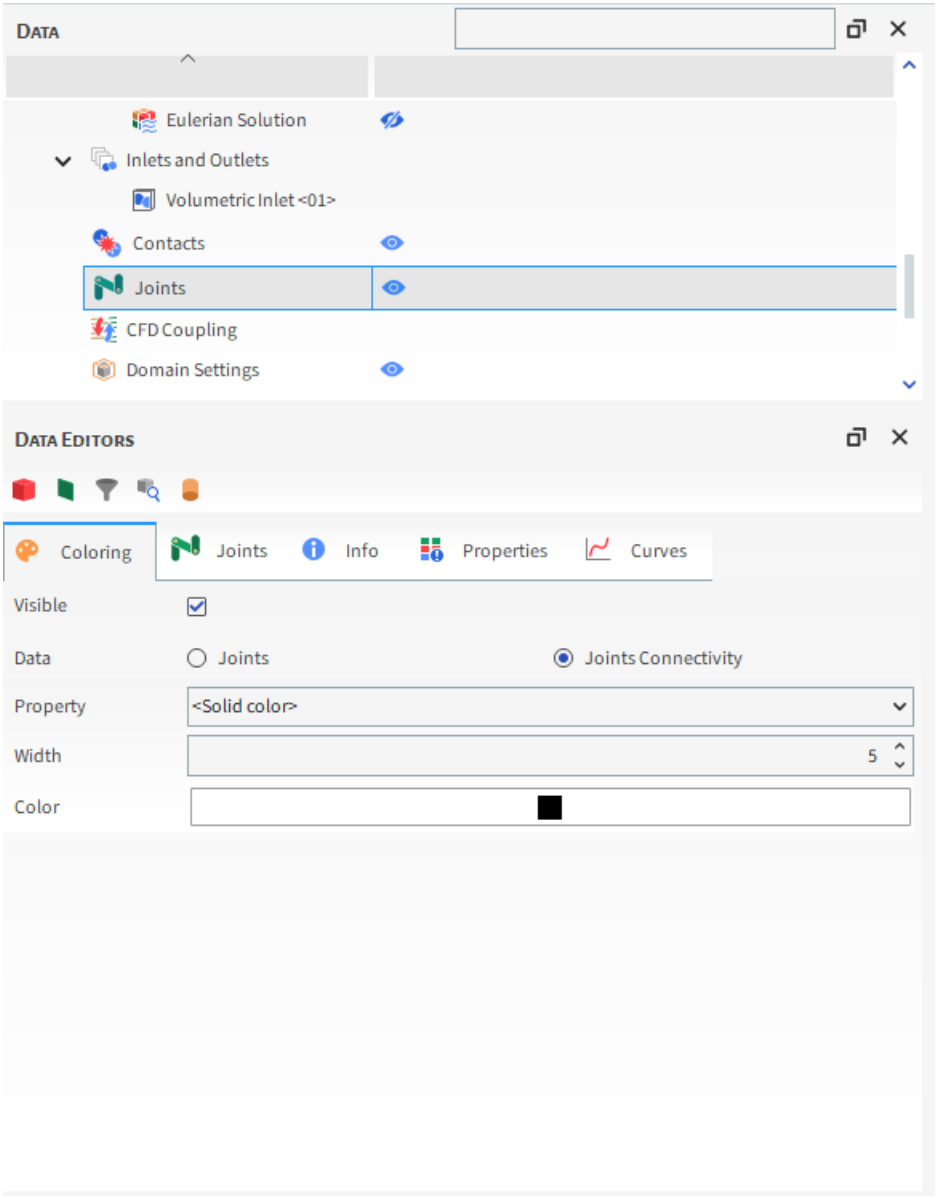


Table 3.1: Joints Parameters - Coloring

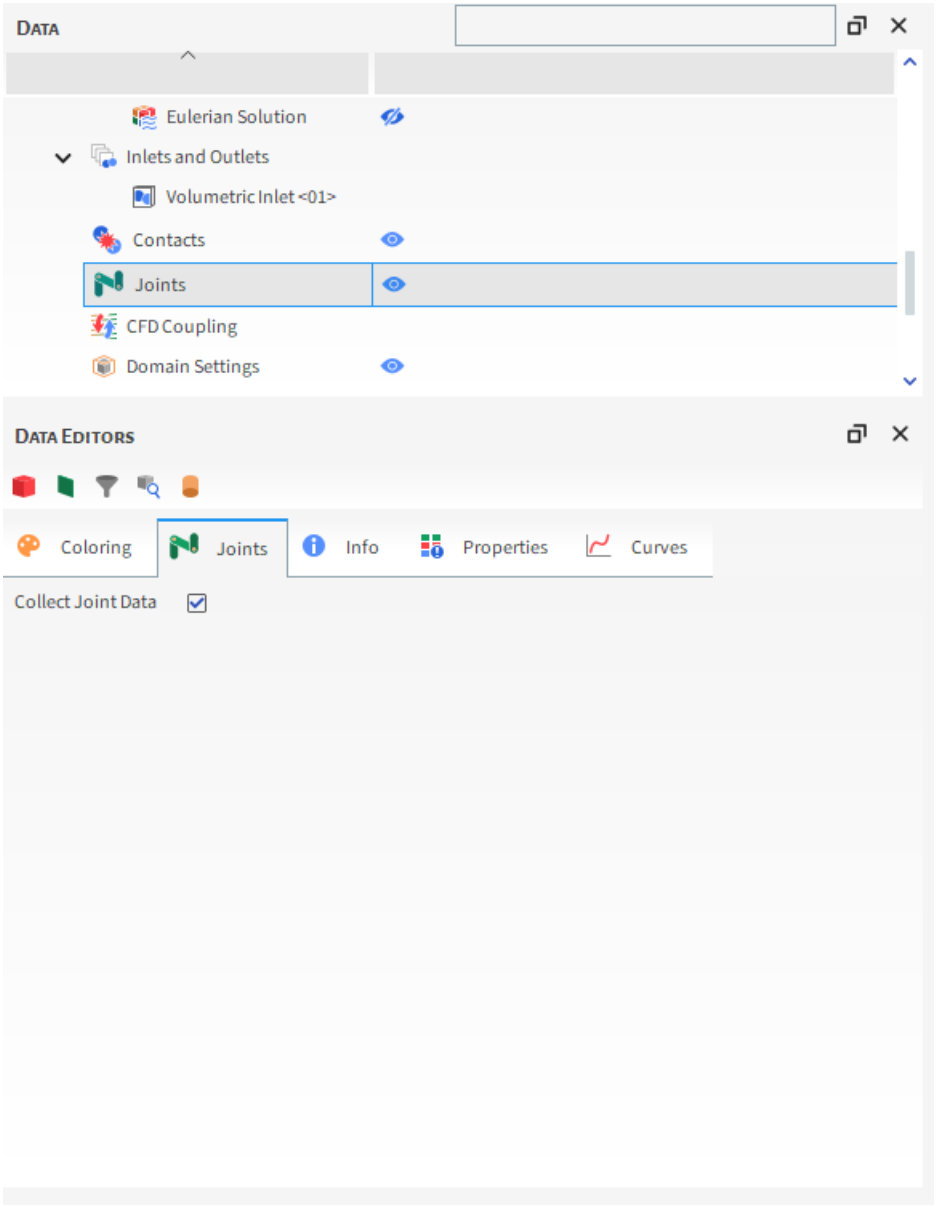
Setting	Description	Range
Visible	When enabled, shows the selected entity in the active view window.	Turns on or off.
Stride	One out of this number of data points will be shown when displaying Joints. The lower the number, the more data points will be displayed.	Whole numbers 1 or greater.
Data	**Joints:** Enables the selected joints to be colored in the 3D View.	Options limited by the selected Data.

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

Setting	Description	Range
	Joints Connectivity: Enables the selected joint connectors to be colored in the 3D View.	
Property	Provide properties and other color options that apply to the display Data type within which the list is contained. • **<Solid Color>** applies a single color to the display Data type. • The remaining properties come directly from the component's Properties tab. Selecting one of these properties colors display Data type in a range according to the data collected for the current time step.	<Solid color>; List of properties automatically generated from the Properties tab.
Point size	When Joints is selected, this changes the size of the dots used to draw the Joints.	Positive Values.
Width	When Joints Connectivity is selected, this changes the thickness of Positive values the lines used to draw the borders.	Positive Values.
Color	When Joints or Joints Connectivity are selected and the Property is **<Solid Color>**, this enables the selected display Data type to be colored in the one solid color chosen.	Options limited by the selected Color.

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

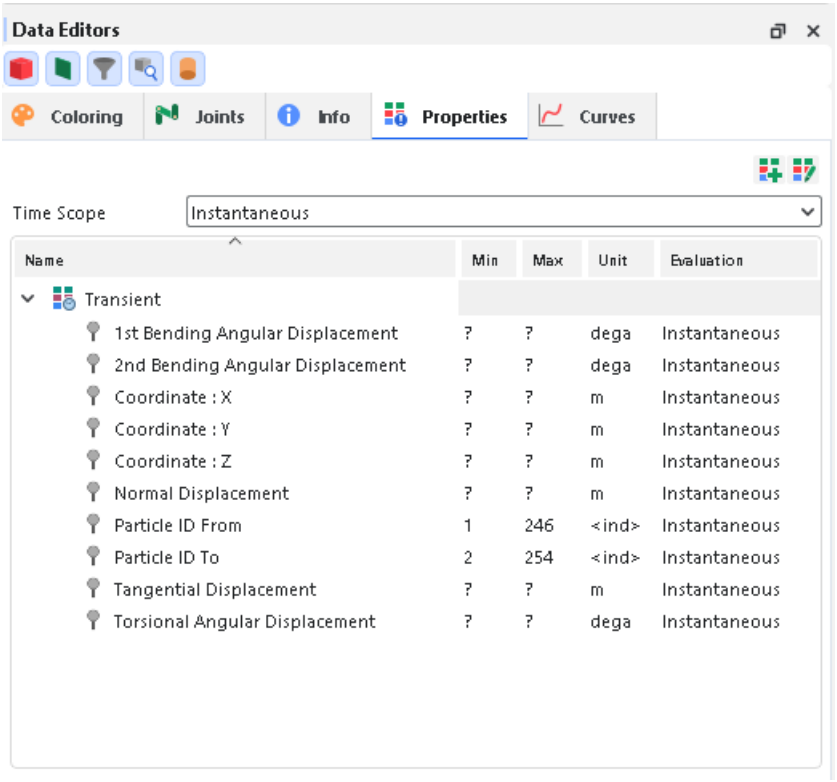
Figure 3.13: Joints parameters in the Data Editors panel (default, unsaved project)



Select **Collect Joint Data** checkbox for joint data to be collected. This way, after simulation, it's possible to analyze joints positions and displacements in properties tab.

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

Figure 3.14: Joints properties in the Data Editors after simulation (default, unsaved project)



Important: To visualize Joints parameters, it's necessary to clear the Meshed Particles Upscaling (p. 1033) checkbox.

See Also:

- About Physics Parameters (p. 417)
- About Modules Parameters (p. 431)
- About Domain Settings Parameters (p. 452)
- About Solver Parameters (p. 464)
- Set Simulation Parameters (p. 414)

3.2.6. Domain Settings Parameters

Domain settings enable you to define Coordinate Limits and, if desired, an additional Periodic Domain for your simulation project.

ABOUT COORDINATE LIMITS

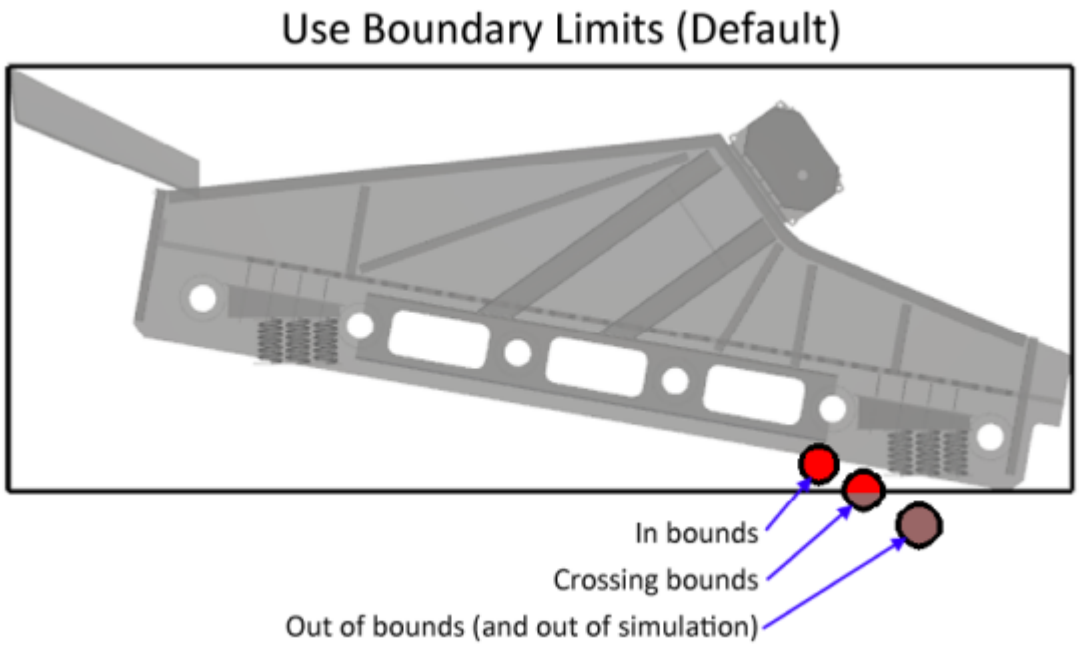
Coordinate Limits determine where in the simulation calculations may occur. This means that any items that fall outside the limits you have defined-including particles and geometries-will not be included in the simulation calculations.

For particles, falling outside the coordinate limits also means they will not be visualized. For particles that are set to be released outside the coordinate limits, this means they will not appear in the simu-

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

lation at all. For particles that were within the limits but then exited out of them, this means those particles will disappear immediately upon leaving the limits (Figure 1).

Figure 3.15: Particle behavior when exiting the coordinate limits



Note: Rocky uses a particle's center point to determine whether it is inside or outside the domain.

For geometries, falling outside the coordinate limits means only that they are not included in calculations. Geometries will still be visualized regardless of whether or not they are within the coordinate limits.

Because of these reasons, it is important that you set your coordinate limits appropriately for the type of simulation you are setting up. For closed systems, such as mixers and mills, where particles are not expected to fall outside of the geometry limits, choosing to use the **Use Boundary Limits** checkbox might be the best option.

For open systems, such as crushers or vibrating screens, where particles are expected to reach beyond the limits of the geometries, setting your own custom coordinate limits might be a better option.

Tip:

To see a walk-through example of setting custom coordinate limits, refer to Tutorial - Vibrating Screen in the *Rocky Tutorial Guide*.

ABOUT PERIODIC DOMAINS

Periodic Domains enable you to include an additional domain within your simulation Coordinate Limits within which particles that are exiting out one side of the periodic domain area are recycled back into the domain from the opposite side.

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

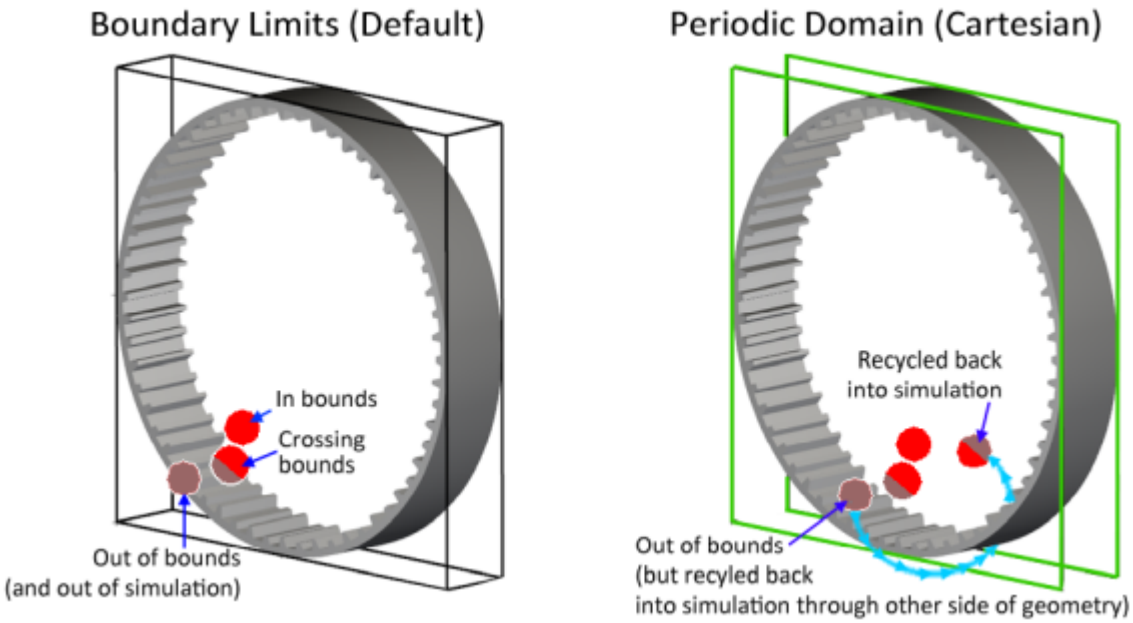
In this version of Rocky, periodic domains can either be **Cartesian**, which are based upon two parallel planes (box-shaped domain), or **Cylindrical**, which are based upon two intersecting planes rotated by an angle to create a cylindrical sector (wedge-shaped domain).

Both types of periodic domains do not allow the injection of particles outside the bounds of the periodic domain. This means that when setting up your Particle Inputs, ensure that your Continuous Injection Inlet, Volume Fill box bounds, or Custom Input definitions are located well within your periodic domain. (See also About Adding and Editing Particle Inputs (p. 708).)

ABOUT CARTESIAN PERIODIC DOMAINS

Cartesian Periodic Domains are the original type of what used to be known as "periodic boundaries" and whose functionality has been included with Rocky since its earliest releases. When choosing to use Cartesian Periodic Domains, two parallel planes are created along the axis (or combination of axes) you specify. Particles that exit the simulation through one plane will reappear in the simulation through the opposite plane (Figure 2).

Figure 3.16: Particle behavior when exiting only the coordinate limits (left) and then with a periodic cartesian domain set (right)



This is especially useful for simulating cross sections or slices of mills; particles flung out of one side of the mill cross section can be recycled back into the simulation from the other side.

Tip:

If you choose to set Cartesian periodic domains, ensure that your periodic domain width (the distance between the **Min Coordinate** and **Max Coordinate** values) is at least 2.5 times wider than your largest particle size. Or, alternatively, ensure that your largest particle size is less than 0.4 times the width of your periodic domain.

For example, a 0.1 m particle can work in a 0.25 m or wider periodic domain. Alternatively, a 1 m periodic domain can support a 0.4 m or smaller particle size. (See also My particle size is too large for my periodic domain (p. 1324).)

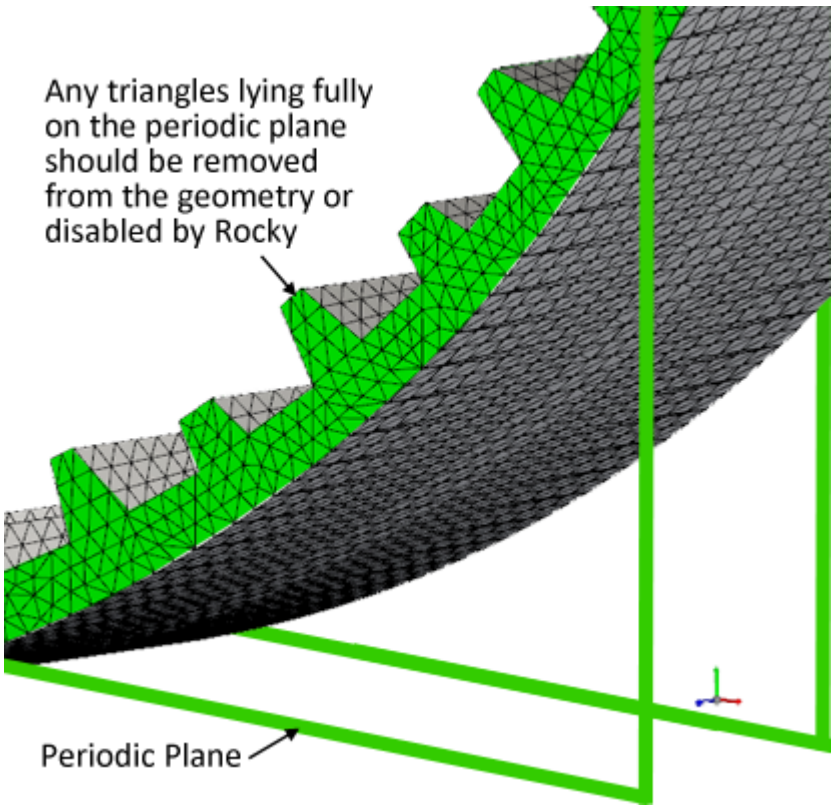
Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

Note: Rocky's Cartesian Periodic Domain functionality was originally designed to be located at the extreme ends of the geometry in question; for example, at the limits of a Mill slice. Sometimes, however, it is desirable to make your periodic domain smaller than the geometry and/or coordinate limits of the domain. Rocky will support you doing this, but be aware that depending on the configuration of the geometries and the periodic limits, it is possible that during the simulation, some particles might go through the walls of your geometries in an unexpected manner. (See also Particles are going through the walls of my geometry (p. 1353).)

Important:

As shown in Figure 3 below, for simulations where the geometry motion is in the periodic plane (i.e., if the periodic direction is Z then the periodic plane is X-Y; if the periodic direction is Y then the periodic plane is X-Z; and if the periodic direction is X then the periodic plane is Y-Z), any triangles whose all 3 vertices exactly lie on the periodic plane should either be removed from the geometry design in a CAD (p. 1389) tool prior to being imported into Rocky, or be disabled by Rocky to ensure proper simulation results. In this version of Rocky, these triangles are disabled for you by default. (See also About Solver Parameters (p. 464).)

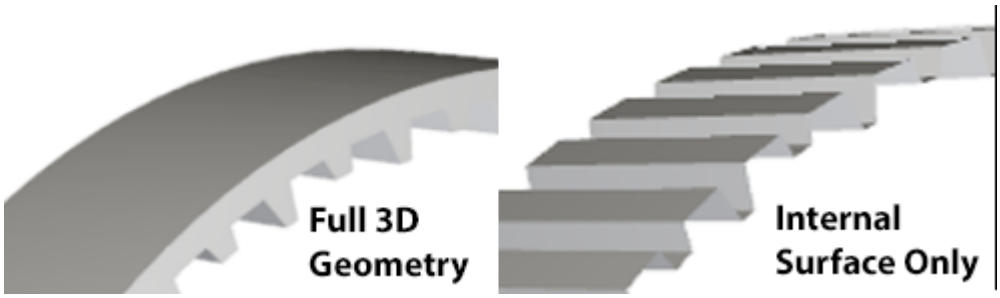
Figure 3.17: Geometry triangles lying fully on the periodic plane (in green) should be removed or disabled



It is recommended that you import only the geometry's internal surface (Figure 4) instead of using the automatic triangle disabling from Rocky. When disabled only, the triangles can be deformed due to wear, and it can cause instabilities in the process — particles could enter into the geometry, for example.

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

Figure 3.18: Comparison between internal surface and Full 3D geometry imports



Tip:

To see a walk-through example using Cartesian Periodic Domains, refer to the following Workshop:

Tutorial - SAG Mill in the *Rocky Tutorial Guide*.

ABOUT CYLINDRICAL PERIODIC DOMAINS

Similar to Cartesian Periodic Domains, Cylindrical Periodic Domains enable you to specify the locations of two planes between which particles are recycled. But rather than being parallel, the two planes in a Cylindrical Periodic Domain are intersecting and rotated by an angle, creating a cylindrical sector, which can look similar in shape to a "wedge" or "slice" of pie.

This can be useful in cylindrical mixing devices, for example, where simulating only a slice of the device would have similar particle properties as simulating the full device, but with many fewer particles and calculations.

The size of the wedge is determined by the number of (evenly spaced) radial divisions you specify for the cylinder. So a division of 2 would create a wedge half the size of the cylinder; a division of 3 would create a wedge a third the size of the cylinder, and so on.

The placement of the wedge is determined both by the location of the cylinder, which is orientated along the X, Y, or Z **Periodic Direction** you specify, and the **Initial Angle** value you specify for the first plane, which will be measured in the positive plane perpendicular to the **Periodic Direction**.

Tip:

The positive angular direction is always counterclockwise when the **Periodic Direction** is set to point towards outside the screen (top view). This means:

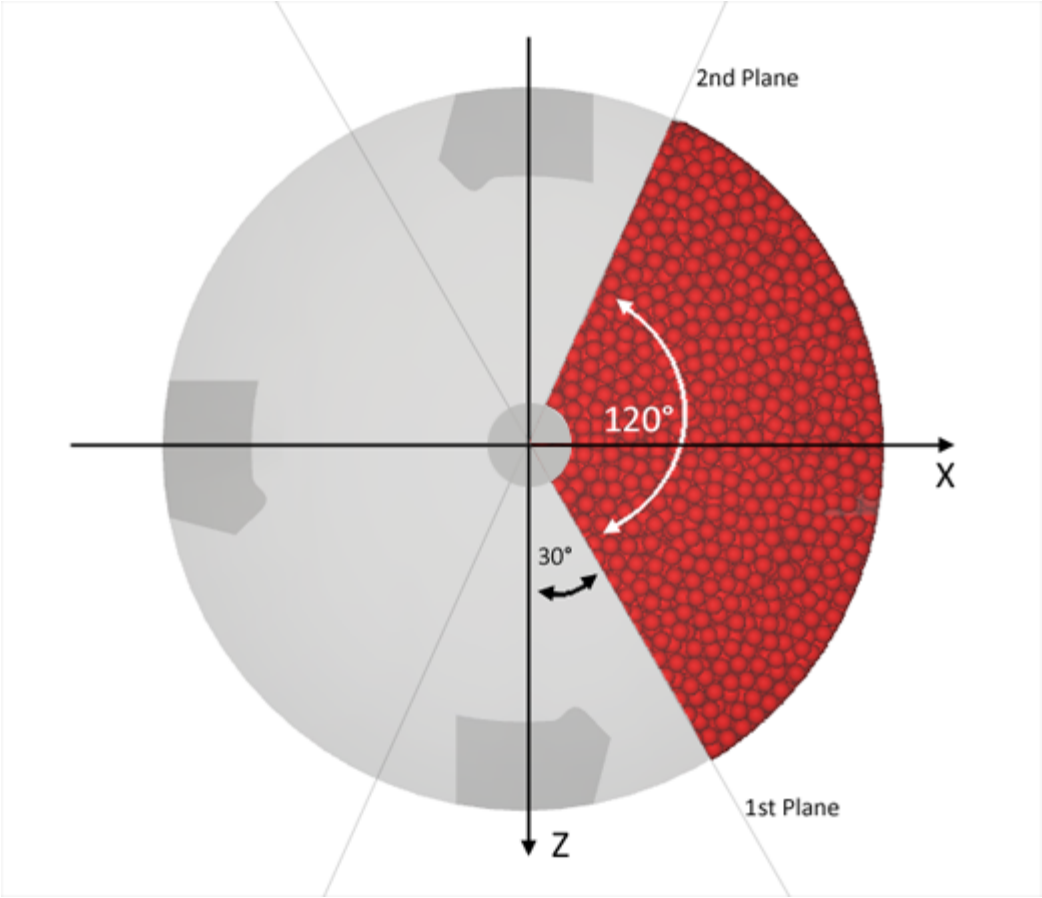
- For a **Periodic Direction** of **Y**, the **Initial Angle** is measured from the **positive Z** axis
- For a **Periodic Direction** of **X**, the **Initial Angle** is measured from the **positive Y** axis
- For a **Periodic Direction** of **Z**, the **Initial Angle** is measured from the **positive X** axis

In the mixing drum example shown in Figure 5, the **Periodic Direction** is Y, the **Initial Angle** is `30 degrees`, and the **Number of Divisions** is 3. These settings result in a periodic, wedge-shaped domain

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

(area in red) that begins 30 degrees counterclockwise in the positive Z direction, and ends 120 degrees (360/3 divisions) later.

Figure 3.19: Illustration of how the size and location of a Cylindrical Periodic Domain is determined



Limitations and Best Practices

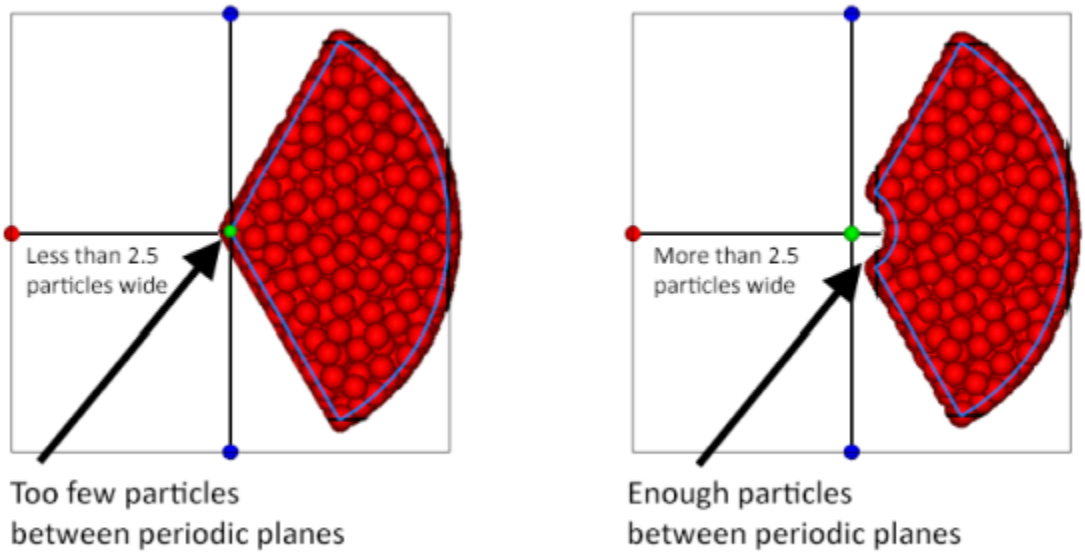
When setting a Cylindrical Periodic Domain, keep in mind the following requirements, limitations, and best practices:

- The **Periodic Direction** must be aligned with one of the three main axes (X, Y, or Z) and must also be parallel to the gravity direction you have set for the simulation.
- Because the size of the wedge is determined by how many (evenly spaced) radial divisions you specify, the **Number of Divisions** value must be a whole integer equal to 2 or greater.
- Your Continuous Injection Inlet, Volume Fill box bounds, or Custom Input definitions must be located within the limits of the periodic domain. (See also About Adding and Editing Particle Inputs (p. 708).)
- Using a **Particle Trajectory** User Process with a Cylindrical Periodic Domain might result in incorrect results. (See also My trajectory lines are outside my periodic domain (p. 1376).)
- It is recommended that you design your geometries, define your particle sizes, and set your Cylindrical Periodic Domain limits such that your periodic domain width (the distance between the periodic planes) at any radial distance from the origin is at least 2.5 times wider than your largest

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

particle size. As illustrated on the left side of Figure 3.20: Cylindrical Periodic Domain with full wedge (left) and with center removed (right) (p. 458), in most cases, particles at the narrowest part of the wedge will not meet these requirements, which is likely to cause instabilities and affect the accuracy of the results. It is therefore recommended that you apply Cylindrical Periodic Domains only to particles that are simulated in a ring shape, similar to when you specify a Cylinder User Process with the **Hole** defined (right side of Figure 3.20: Cylindrical Periodic Domain with full wedge (left) and with center removed (right) (p. 458)).

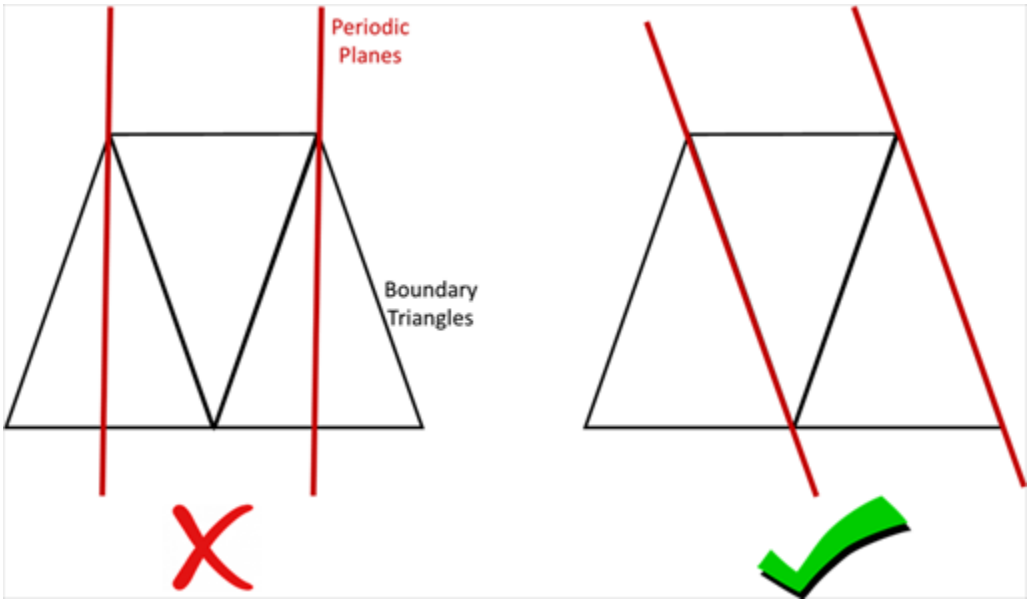
Figure 3.20: Cylindrical Periodic Domain with full wedge (left) and with center removed (right)



- To reduce the chance of simulation errors, ensure that your geometry triangles fall inside the selected domain and that the periodic planes do not cross the triangle surfaces. The ideal situation is to have the periodic planes crossing along the exact edges of the triangles, as shown on the right side of Figure 3.21: Periodic planes intersecting boundary triangles (left) and aligning on the edges (right) (p. 459).

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information
of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

Figure 3.21: Periodic planes intersecting boundary triangles (left) and aligning on the edges (right)



DOMAIN SETTINGS PARAMETER DEFINITIONS

Use the figures and table below to help you understand the various Domain Settings parameters you can set for a simulation project.

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

Figure 3.22: Domain Settings, Coordinate Limits parameters in the Data Editors panel (No periodic domain)

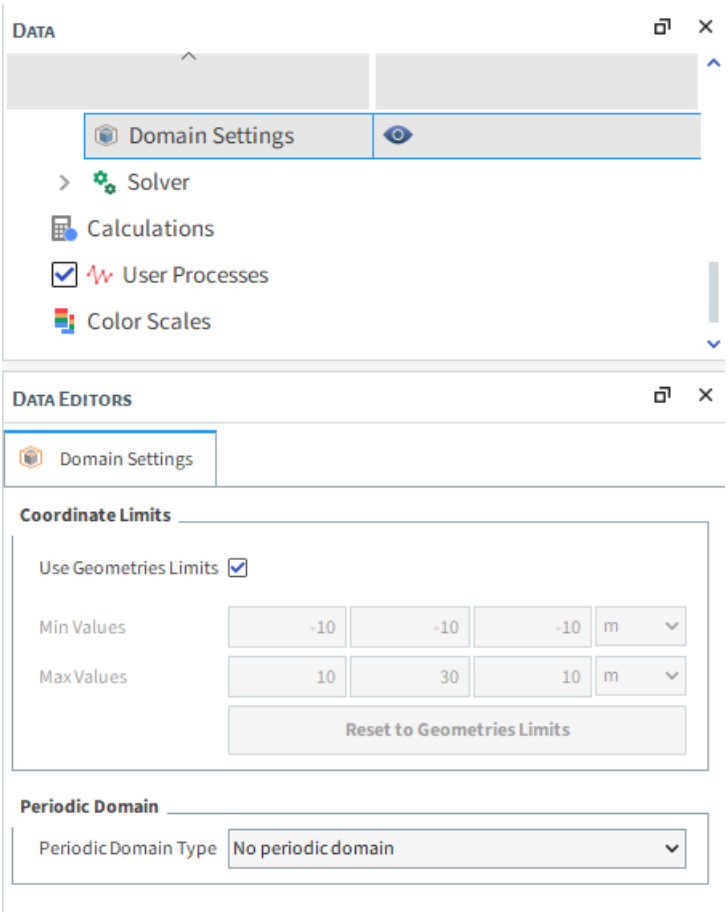


Figure 3.23: Domain Settings, Cartesian Periodic Domain Type parameters

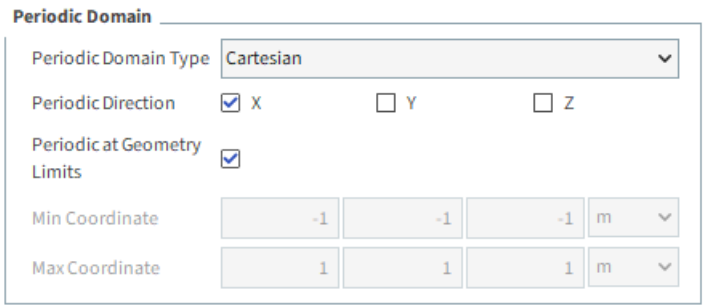
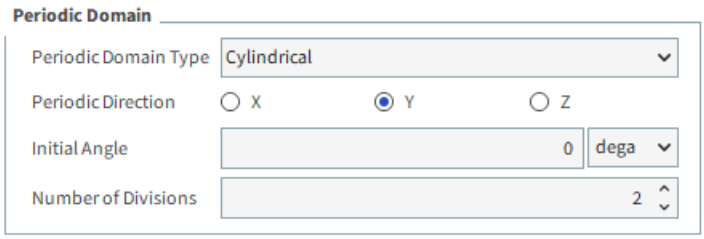


Figure 3.24: Domain Settings, Cylindrical Periodic Domain Type parameters



Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

Table 1: Domain Settings parameter options

Setting	Description	Range
Coordinate Limits		
Use Geometries Limits	When selected, automatically sets the coordinate limits of the simulation at the extreme ends of the existing geometries. When geometries change or move, the limits will be changed also. Notes: • Selecting this item will cause particles to disappear from the simulation when they reach the end of a geometry. • Clearing this item will cause particles to disappear from the simulation based on the values you set in the **Min Values** and **Max Values** fields. • Because the limits are dynamic due to the changing and moving of geometries, the limits will not be visible in the 3D View. You can, however, review the boundary limit values by viewing the **Simulation Summary** screen. (See Verify Key Simulation Criteria in the Summary (p. 858) for more information.)	Turns on or off
Min Values	When **Use Geometries Limits** is cleared, this enables you to set the minimum values for the simulation coordinate limits in the following format: *X Y Z* These custom limits will be visible in the 3D View.	No limit for X, Y, and Z values but must be lower than **Max Values**
Max Values	When **Use Geometries Limits** is cleared, this enables you to set the maximum values for the simulation coordinate limits in the following format: *X Y Z* These custom limits will be visible in the 3D View.	No limit for X, Y, and Z values but must be higher than **Min Values**
Reset to Geometries Limits	When the **Use Geometries Limits** option is disabled, **Reset to Geometries Limits** allows you to automatically set the Domain's minimum and maximum values to the limits of the current geometries.	
Periodic Domain		

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

Setting	Description	Range
Periodic Domain Type	Enables you to choose whether or not to include a periodic domain, and if so, what type of domain you want to include. Specifically: • **No periodic domain** will use only the **Coordinate Limits** to define the domain. • **Cartesian** enables you to include a periodic domain defined by two parallel planes. • **Cylindrical** enables you to include a periodic domain defined by two intersecting planes rotated by an angle, which creates a cylindrical sector or wedge shape.	No periodic domain; Cartesian; Cylindrical
Periodic Direction	When either **Cartesian** or **Cylindrical** is selected as **Periodic Domain Type**, this determines the direction periodic domains are enabled. Specifically: • For **Cartesian**, two parallel planes will be implemented along the X, Y, or Z axis (or combination of axes) you specify. • For **Cylindrical**, the cylinder for which the two intersecting planes are based is aligned along either the X, Y, or Z axis you specify. Important: the direction you select here must match the gravity direction for the simulation.	Turns on or off
Periodic at Geometry Limits	When **Cartesian** is set for **Periodic Domain Type**, enabling this checkbox locates the periodic domain at the farthest edge of the simulation geometries. When cleared, the **Min. Coordinate** and **Max. Coordinate** values will be used to define the periodic domain limits.	Turns on or off
Min Coordinate	Location along the axis specified for **Periodic Direction** to place the first of two planes that define the box-like shape of the cartesian periodic domain limits. **Tip:** Ensure the distance from this coordinate to the **Max Coordinate** is at least 2.5 times wider than the largest particle size.	No limit

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

Setting	Description	Range
Max Coordinate	Location along the axis specified for **Periodic Direction** to place the second of two planes that define the box-like shape of the cartesian periodic domain limits. **Tip:** Ensure the distance from this coordinate to the **Min Coordinate** is at least 2.5 times wider than the largest particle size.	No limit
Initial Angle	When **Cylindrical** is set for **Periodic Domain Type**, this determines the location of the first of two intersecting planes that define the wedge-like shape of the cylindrical periodic domain limits. How this value is measured depends upon the **Periodic Direction** specified. Specifically: • A **Periodic Direction** of **Y** will measure the **Initial Angle** from the **positive Z** axis. • A **Periodic Direction** of **X** will measure the **Initial Angle** from the **positive Y** axis. • A **Periodic Direction** of **Z** will measure the **Initial Angle** from the **positive X** axis. Entering a 0 (zero) value will align the first plane with the axis of measure; entering any other value will offset the plane by that angle amount.	No limit
Number of Divisions	When **Cylindrical** is set for **Periodic Domain Type**, this defines how the cylinder will be divided to determine the final size of the wedge. Specifically: • A value of 2 will create a wedge equal to half the cylinder. (The second plane will be placed 180 degrees offset from first plane.) • A value of 3 will create a wedge equal to a third of the cylinder. (The second plane will be placed 120 degrees offset from first plane.) • A value of 4 will create a wedge equal to a quarter of the cylinder. (The second plane will be placed 90 degrees offset from first plane.) • (And so on.)	Whole numbers greater than or equal to 2

See Also:

• About Study Parameters (p. 415)

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

- About Physics Parameters (p. 417)
- About Modules Parameters (p. 431)
- About Solver Parameters (p. 464)
- Set Simulation Parameters (p. 414)
- My trajectory lines are outside my periodic domain (p. 1376)

3.2.7. Solver Parameters

Unlike most setup parameters, which you can define only prior to processing, you are able to change the parameters under **Execution** on the **Solver** | **General** tab, or any of the options on the **Solver | Advanced** tab after processing has started without invalidating your results. You must first **Stop** processing, change the parameters you want, and then **Resume** processing again in this scenario. (See also I cannot change my setup parameters during processing (p. 1361).)

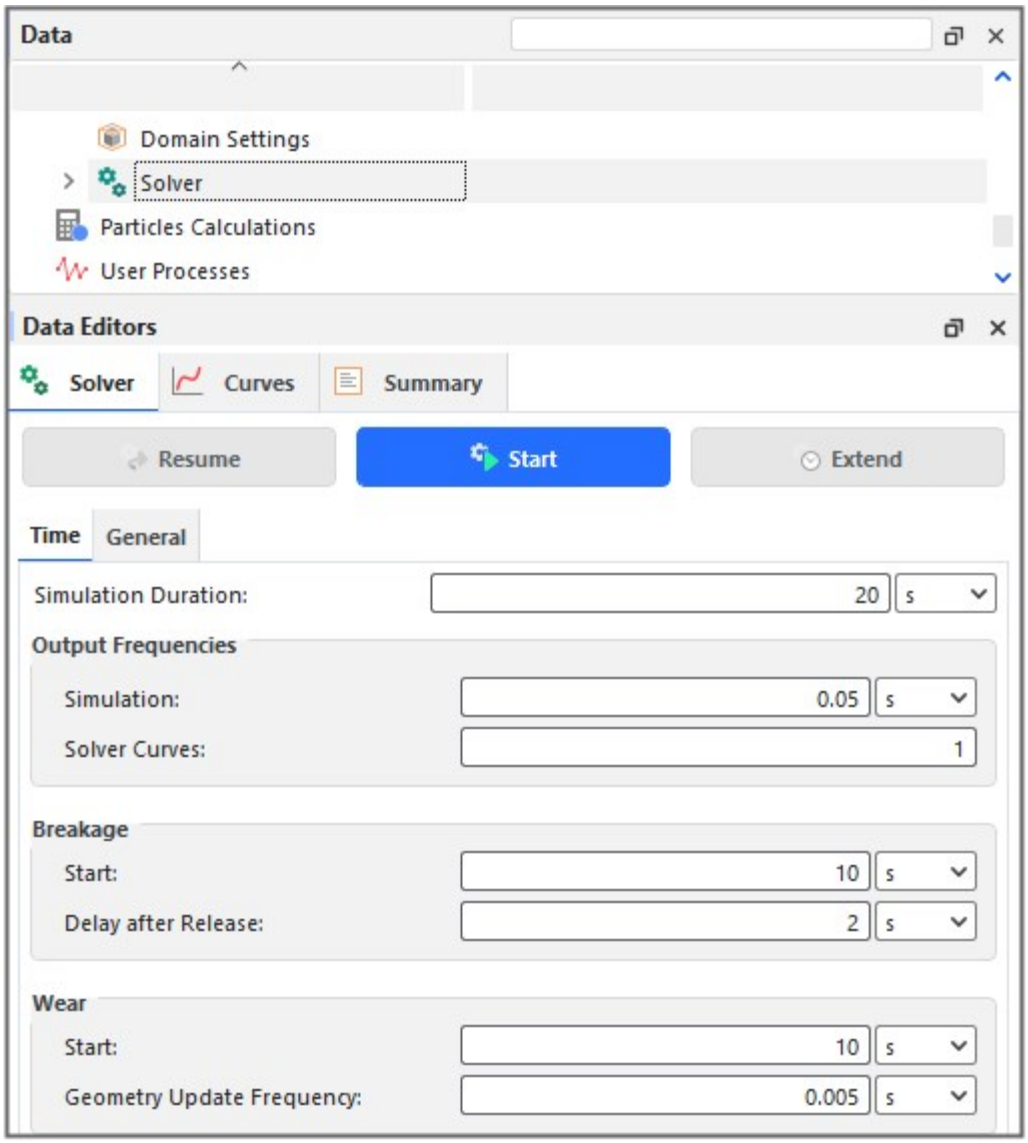
Important:

In this version of Rocky, Particles Energy Spectra and Contacts Energy Spectra are defined in their own respective modules. (See also About Particles Energy Spectra (p. 358) and About Contacts Energy Spectra (p. 350).)

Use the figures and tables below to help you understand the various Solver parameters you can set for a simulation project.

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

Figure 3.25: Solver | Time (default) parameters in the Data Editors Panel



Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

Figure 3.26: Solver | Time parameters for 2-Way Fluent CFD Coupling simulations in the Data Editors panel

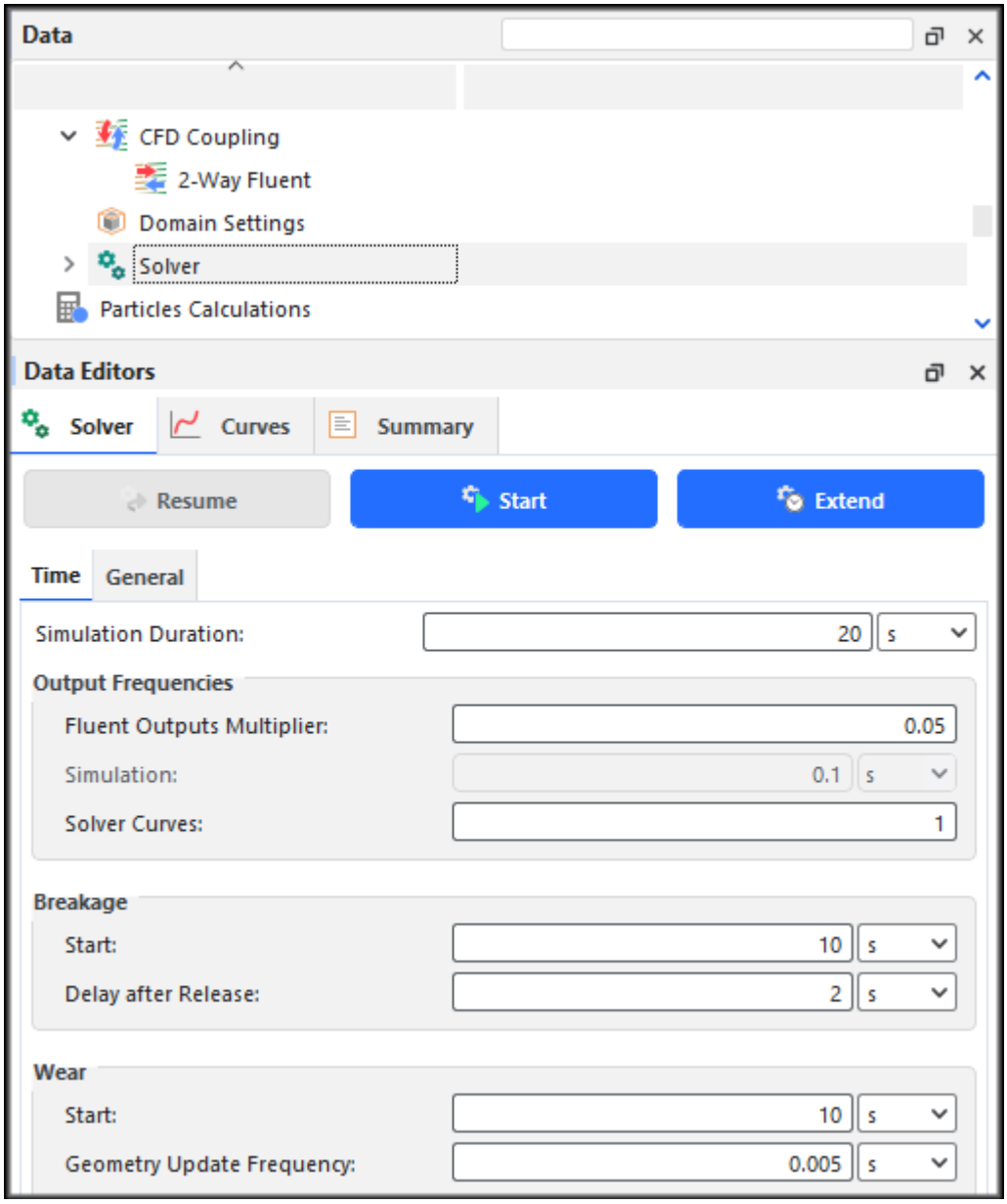


Table 1: Solver | Time parameter options

Setting	Description	Range
Simulation Duration	The total amount of real time that you want the simulation to run. **Tip:** When calculating the **Simulation Duration** value, be sure to account for the length and speed of your conveyors, the mass flow rate of your particles, steady-state, and so on.	Positive values

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

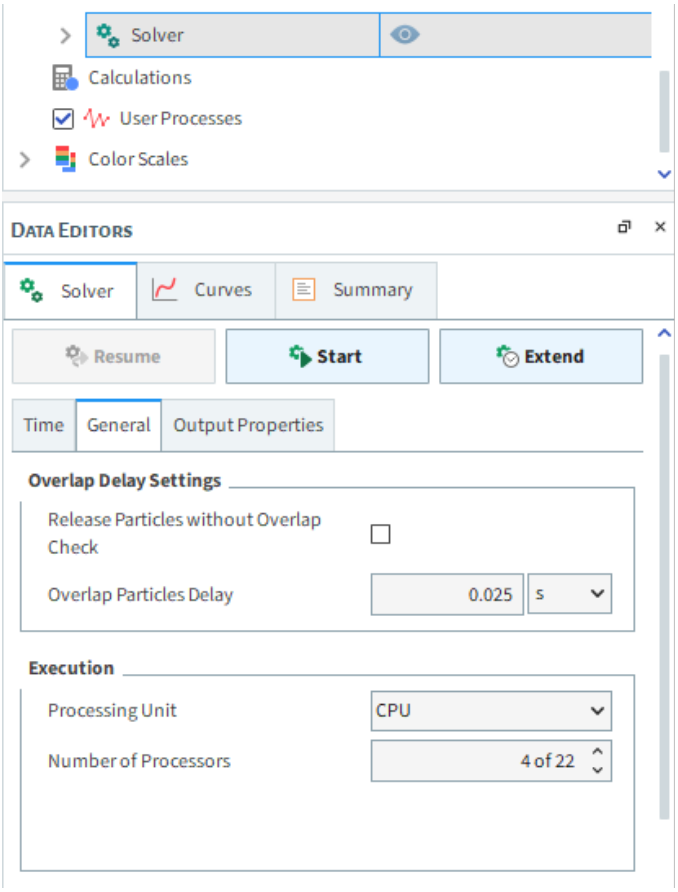
Setting	Description	Range
Output Settings Fluent Outputs Multiplier	For 2-Way Fluent CFD Coupling simulations only, this value determines how many Fluent time steps must occur before simulation files from both Rocky and Fluent are saved. For example, a value of 1 means files will be saved at every Fluent time step; a value of 10 means that files will be saved for every ten Fluent time steps, and so on. (See also About Using the 2-Way Fluent Method (p. 795).)	Positive, whole values
Output Settings Simulation	The time intervals at which you want your *output files* (p. 1393) to be saved. **Note**: This parameter is disabled in 2-Way Fluent CFD Coupling simulations and is instead calculated based upon the Fluent time step and the **Fluent Outputs Multiplier**. (See also About Using the 2-Way Fluent Method (p. 795).) **Tip:** To prevent rotating or vibrating boundaries from appearing like they are moving backwards, divide the rotational velocity (for rotating boundaries) or frequency (for vibrating boundaries) by 2 and then set your simulation output frequency as slightly lower than that value. **Tip:** Rocky also has a feature to select multiple output frequencies during the simulation, which is useful to increase or reduce the output data after some time, so that specific phenomena can be evaluated with more or less detail. (See also I cannot find the Output Frequency parameter (p. 1336).)	Value must be positive but less than **Simulation Duration**
Output Settings Solver Curves	Controls the frequency at which the Solver Curves are updated while the simulation is being processed. The value entered here represents the number of times the Solver Curves will be updated between two consecutive outputs. For example, if the Simulation Output Settings is set to 1 [s] and the Solver Curves Output Frequency is set to 100, it means that the Solver Curves will be updated at every 0.01 [s].	Whole values greater than zero

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

Setting	Description	Range
	Note: Rocky will limit the time interval for updating the Solver Curves Frequency to be larger than 100 simulation time steps.	
Breakage Start	The amount of time you want to wait before starting to calculate particle breakage. **Tip:** It is best practice to set your Breakage **Start** time to begin after a steady state has been reached in your particle flow.	Value must be positive but less than **Simulation Duration**
Breakage Delay after Release	The amount of time you want to wait after a particle has been released before starting to calculate particle breakage.	Value must be positive but less than **Simulation Duration**
Wear Start	The amount of time you want to wait before starting to calculate belt and boundary wear. **Tip:** It is best practice to set your Wear **Start** time to begin after a steady state has been reached in your particle flow.	Value must be positive but less than **Simulation Duration**
Wear Geometry Update Frequency	Amount of time between wear geometry updates. A smaller value will produce smoother transitions between updates but might increase processing time; a larger value will produce rougher transitions between updates, but will take less time to process. (See also See Surface Wear on the Geometry Itself (p. 378).)	Positive

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

Figure 3.27: Solver | General parameters in the Data Editors panel



Note: The Execution options are covered in the About Starting a Simulation section.

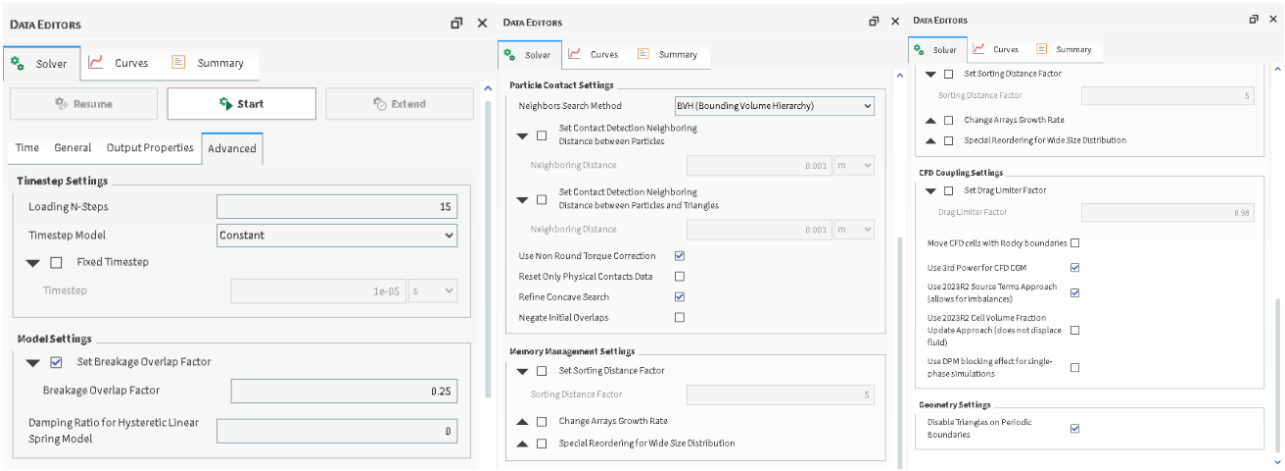
Table 2: Solver | General parameter options

Setting	Description	Range
Release Particles without Overlap Check	For Continuous Injection Inputs only, when enabled, this overrides the setting that delays particles from discharging from an inlet if other particles or boundaries are in the way. When cleared, Rocky will delay releasing any overlapped particles by the **Overlap Particles Delay** time provided. WARNING: Selecting this item with large particles or high particle velocities could cause problems with your simulation and might result in Rocky hanging or shutting down. **Tip:** Rather than releasing particles without checking for overlaps (i.e., enabling this checkbox), it is recommended that you instead increase the duration between your input's **Start Time** and **Stop Time** settings (see also About Adding and Editing Particle Inputs (p. 708)) and/or increase the size of your inlet	Turns on or off

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

Setting	Description	Range
	geometry to allow more space and time for your particles to be released.	
Overlap Particles Delay	When the **Release Particles without Overlap Check** checkbox is cleared, this defines the amount of time Rocky will wait to release a particle from an inlet should it determine that other particles or boundaries are in the way.	Positive values

Figure 3.28: Solver | Advanced tab (Advanced Features) settings in the Data Editors panel



Note: The **Advanced** tab only appears when the **Advanced Features** checkbox is enabled on the **Additional Features** tab of the **Options | Preferences** dialog. (See also About Setting Global Preferences (p. 227).)

Table 3: Solver | Advanced tab (Advanced Features) settings

Setting	Description	Range
Time Step Settings Loading N-Steps	Value used to calculate time steps. In general, the lower the value, the faster the processing but the more unstable the results. Higher values (for example, above 15) are recommended if you are interested in collecting energy spectra or calculating breakage.	Positive values Recommended: 10-30
Time Step Settings Time Step Model	Determines how Rocky treats the calculation of an overlap in cases when a specific collision would allow for a bigger time step than the one defined for the simulation. Important: What you set for **Time Step Model** does not change the time step itself, only whether overlaps are calculated always with Rocky's normal, full procedure (**Constant**) or, whenever possible, with a simplified one (**Variable**). Specifically: • **Constant**: Even for collisions allowing a time step bigger than the one set for the simulation, overlap calculations	Constant; Variable

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

Setting	Description	Range
	are done in full. For *shaped particles* (p. 1395) especially, always using the full calculation can be computationally expensive and might result in slower processing. • **Variable**: For collisions allowing a time step bigger than the one set for the simulation, full overlap calculations are *mostly* skipped (see note below) and a simpler, approximate overlap calculation is used instead. This simplification is adaptive (uniquely set per qualifying collision) and made during a time lapse approximately equal to the allowed time step for the collision. Using these simpler overlap calculations can speed-up processing, especially for shaped particles. **Note**: To ensure that precision is not lost because of this simpler method, the full overlap calculation is still made periodically for those collisions that allow it. *For example*, if the simulation time step is Δt and a collision would allow a time step equal to $n\Delta t$, then the collision starts with a full calculation, followed by $n-1$ sets of simplified calculations. This is followed by a full calculation, then $n-1$ simplified ones, and so on.	
Time Step Settings Fixed Time Step	When enabled, it allows you to define an initial value for the **Time Step Duration**, which is shown on the **Simulation Summary** screen. (See also About the Simulation Summary (p. 859).) When cleared, Rocky will automatically calculate this value based upon particle size, density, elastic properties, and so on. Useful only when you want to override the Rocky value, which could lead to faster processing but could also decrease stability.	Turns on or off
Time Step Settings Time Step	When **Fixed Time Step** is enabled, this sets a value for **Time Step Duration** that will override the value Rocky typically calculates automatically based upon several different factors. (See also About the Simulation Summary (p. 859).)	Positive values
Model Settings Set Breakage Overlap Factor	When enabled, it allows you to define a factor for particle breakage overlap as a fraction of **Minimum Size** (see also About Adding and Editing Particle Sets (p. 602)) of the breakage fragment, which affects how easily particles break when coming into contact with boundaries. When enabled, a particle will break when the **Breakage Overlap Factor** with the boundary it comes into contact with is reached-regardless of the particle's actual strength. The fragment size distribution for this breakage will be defined by the energy of the particle's contact at the time of breakage. **Note**: This overlap factor is enabled automatically by Rocky when Experimental Features (see also About Setting Global Preferences (p. 227)) is turned off. Turning	Turns on or off

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

Setting	Description	Range
	on Experimental Features allows you the ability to change or disable the overlap factor value. When cleared, a contacting particle will break only when its energy strength is reached, thus creating a possibility of the fragments being generated on the other side of a boundary if the overlap on breakage is high enough for that. Enabling this parameter can be useful when you want to avoid fragment generation on the other side of a boundary. This can occur when particles that have large boundary overlaps due to low contact stiffness, break and the fragments in the overlap area end up on the wrong side of the boundary. It can also, however, introduce errors in particle strength distribution.	
Model Settings Breakage Overlap Factor	When **Set Breakage Overlap Factor** is enabled, a particle will break when its overlap with the contacting boundary reaches the value of this factor multiplied by the **Minimum Size** (see also About Adding and Editing Particle Sets (p. 602)) of the fragment-even if the current energy is not enough for the particle to break.	Positive values
Model Settings Damping Ratio for Hysteretic Linear Spring Model	Enables you to specify the value of the damping ratio, η, a dimensionless parameter that is used in the damping coefficient equation defined for the Hysteretic Linear Spring Normal Force model. The damping force in the hysteretic model is only activated during long-term contacts with the singular purpose of eliminating spurious low-amplitude oscillations in the overlap that usually arises in those cases. See also the Rocky DEM Technical manual. (From the **Help** menu, point to **Manuals** and then click **DEM Technical Manual**.)	0-1
Particle Contact Settings Neighbors Search Method	Enables you to specify the method that the solver uses to search neighboring particles. This affects the efficiency and consequently the time and computational resources required to complete a simulation. Each method is best suited for a specific situation. **BVH (Bounding Volume Hierarchy)**: method best suited for higher *PSD* (p. 1394) and high aspect ratio particles. **Regular Grid (Performance Enhanced)**: method best suited for lower *PSD* (p. 1394) and spherical particles. Its focus is on performance, so it may require more computational resources (ex.: system memory). **Regular Grid (Memory saving)**: method best suited for lower *PSD* (p. 1394) and spherical particles. Its focus is on saving computational resources. This computational resources. It	BVH (Bounding Volume Hierarchy); Regular Grid (Performance Enhanced); Regular Grid (Memory Saving)

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

Setting	Description	Range
	allows for cases with more particles to be evaluated, at the expense of the time required to perform the simulation.	
Particle Contact Settings Set Contact Detection Neighboring Distance between Particles	When enabled, it allows you to manually set the distance away from a particle that defines the boundary within which *neighboring particles* (p. 1392) are considered for contact. When cleared, this distance is automatically calculated by Rocky.	Turns on or off
Particle Contact Settings Neighboring Distance	When **Set Contact Detection Neighboring Distance between Particles** is enabled, this value overwrites the distance that is automatically calculated by Rocky.	Positive values
Particle Contact Settings Set Contact Detection Neighboring Distance between Particles and Triangles	When enabled, it allows you to manually set the distance away from a particle that defines the boundary within which *neighboring triangles* (p. 1393) are considered for contact. When cleared, this distance is automatically calculated by Rocky.	Turns on or off
Particle Contact Settings Neighboring Distance	When **Set Contact Detection Neighboring Distance between Particles and Triangles** is enabled, this value overwrites the distance that is automatically calculated by Rocky.	Positive values
Particle Contact Settings Use Non Round Torque Correction	When enabled for *shaped particles* (p. 1395), this improves the contact overlap algorithm by increasing the accuracy of the contact point position calculations and thus increasing the accuracy of the rotational calculations for shaped particles. Notes: • In this version of Rocky, this setting is enabled by default; however, for projects simulated in versions of Rocky prior to v4.1.0, note that this algorithm was not yet implemented. • Enabling this setting is expected to increase simulating processing time by a small percentage (< 3%).	Turns on or off

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

Setting	Description	Range
Particle Contact Settings Reset Only Physical Contacts Data	When enabled for *shaped particles* (p. 1395), the contact parameters during contact detection will be reset from existing ones for physical contact only. When cleared, both near contacts and physical contacts will result in a reset. Enabling this setting will lead to slower calculations but will save memory, which might be important for GPU simulations in particular.	Turns on or off
Particle Contact Settings Refine Concave Search	When enabled, Rocky performs a second, more refined search for *neighboring particles* (p. 1392) in addition to the standard search that is done by default. This second search is specifically designed for concave shapes, and serves to provide a more accurate near-neighbors list by removing particles from the initial results that are not actually near neighbors. Reducing the results for concave shapes saves memory and may also save computational time due to fewer possibilities for contacts needing to be considered. For convex shapes processed on GPUs, however, this may slow down processing so it is recommended that this be enabled only for concave shapes. When cleared, only the standard search method is used.	Turns on or off
Particle Contact Settings Negate Initial Overlaps	Especially for projects in which some particles are enabled only after a certain condition is met (distance, temperature, time, etc.), this feature can help you avoid unrealistic forces due to a just-enabled particle landing inside the boundaries of another particle. When enabled, Rocky negates—for only the first collision a particle experiences— the first overlaps. This means that when a particle is enabled later into the simulation and happens to land within the boundaries of another particle, those initial overlaps will not be computed. It is only on the subsequent overlaps that contact forces detection will start. When cleared, overlaps are always considered in forces calculations. Notes: • Here, an *enabled particle* refers to a particle that was previously not active due to being *frozen* (for example, from the **Particle Freeze Outside Box** module) or having its *collisions disabled* (for example, from the **Disable Collision Outside Region** module). • If you do not already have sign-in information for the customer portal, please contact your Rocky representative for assistance.	Turns on or off

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

Setting	Description	Range
Memory Management Settings Set Sorting Distance Factor	When enabled, allows you to control how frequently particle data stored in memory is reordered in a sorting operation. (This operation happens on both CPU and GPU). Sorting helps guarantee that data from a particle is stored in memory near the data from *neighboring particles* (p. 1392) in the 3D space, which can help with performance. The sorting distance itself is defined in Rocky as the average distance between neighbor particles times the **Sorting Distance Factor**, the latter of which this parameter enables. During the simulation, whenever a particle moves a distance larger than or equal to the sorting distance since the last sorting, a new sorting operation is triggered for all active particles at that time. Since sorting can be relatively expensive in terms of computation time, an adequate sorting frequency may be crucial for improving performance in cases where particles are moving at high velocities. When cleared, the sorting distance calculated by Rocky is used.	Turns on or off
Memory Management Settings Sorting Distance Factor	When **Set Sorting Distance Factor** is enabled, this value is multiplied by the average distance between neighbor particles calculated by Rocky. The larger the value of this factor, the less frequent the sorting.	Positive values
Memory Management Settings Change Arrays Growth Rate	When enabled, it allows you to change the rate of growth for the internal memory arrays used during simulation processing. When cleared, the default value of 1.0 will be used.	Turns on or off
Memory Management Settings Arrays Growth Rate	When **Change Arrays Growth Rate** is enabled, this allows you to change the rate of growth for the internal memory arrays used during simulation processing.	Positive values
Memory Management Settings Special Reordering for Wide Size Distribution	When enabled, bigger particles are moved to the end of the internal memory array used to store them during processing. This reordering can be especially useful in cases with big differences in particle sizes (size ratios ~ 100 for instance), which can sometimes cause memory issues during processing. When cleared, only the standard order is used.	Turns on or off
Memory Management Settings Particle Size Limit for Reordering	When **Special Reordering for Wide Size Distribution** is enabled, this defines the minimum size of the "bigger" particles in the simulation that will be reordered.	Positive values **Note**: This value must be smaller

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

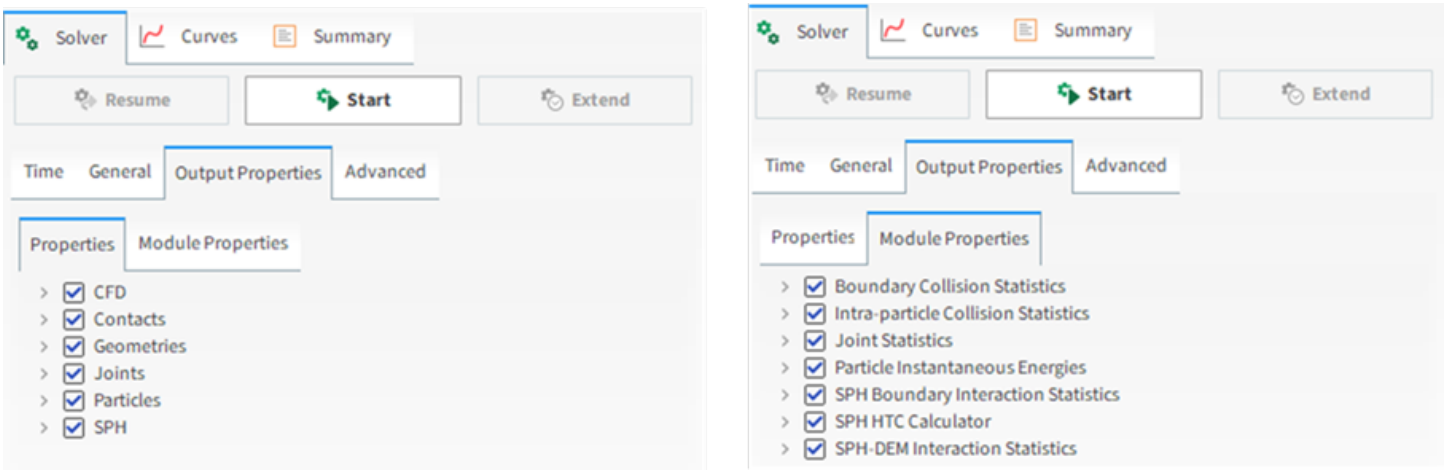
Setting	Description	Range
		than the size of the biggest particle in the simulation
CFD Coupling Settings Set Drag Limiter Factor	When enabled, it allows you to limit the effects of drag upon the particles that are affected by fluid interactions. When cleared, Rocky uses the full force of the **Drag Law** that is set. (See also About Using the 1-Way Fluent Method (p. 775) and About Using the 2-Way Fluent Method (p. 795).)	Turns on or off
CFD Coupling Settings Move CFD Cells with Rocky Boundaries	It uses motion properties in order to rotate the fluid mesh inside Rocky, so that both CFD domain and Rocky boundaries are always syncronized. This allows the use of larger CFD time steps, resulting in improved performance and a stable solution. When enabled it seamlessly synchronizes the movement of the Fluent CFD mesh with the corresponding movement of the Rocky boundaries, ensuring smooth coordination between them. **Note:** This feature is available only for 2-way CFD coupled simulations with moving meshes.	Turns on or off
CFD Coupling Settings Drag Limiter Factor	When **Set Drag Limiter Factor** is enabled, the drag applied to the particles is limited by this value, f, times the relative velocity, v, times the particle mass,m: $\frac{fvm}{dt}$ Setting values below 1 increases the stability of integration for very small particles with large velocities relative to the fluid. However, it also decreases the accuracy of drag force calculations.	Positive values
CFD Coupling Settings Use 3rd Power for CFD CGM	When it is enabled the forces will be corrected by a third power factor instead of the default second power.	Turns on or off
CFD Coupling Settings Use 2023 R2 Source Terms Approach	When it is enabled the 2023 R2 Source Terms Approach will be used in 2-Way couplings simulations, allowing for imbalances.	Turns on or off

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

Setting	Description	Range
(allows for imbalances)		
CFD Coupling Settings Use 2023 R2 Cell Volume Fraction Update Approach (does not displace fluid)	When it is enabled the 2023 R2 Cell Volume Fraction Update Approach will be used.	Turns on or off
CFD Coupling Settings Use DPM blocking effect for single phase simulations	When it is enabled the DPM blocking effect will be considered for 2-Way single phase simulations.	Turns on or off
Geometry Settings Disable Triangles on Periodic Boundaries	When enabled for simulations using Cartesian Periodic Domains (see also About Domain Settings Parameters (p. 452)), this option disables from calculations (but keeps in visualizations) any geometry triangles whose all 3 vertices land exactly on the periodic domain. Because of the way periodic domains work, it is best practice to remove these triangles when initially designing your geometry in a *CAD* tool. This option ensures that if any of these triangles do appear in your simulation, they will not negatively affect your calculations. When cleared, any geometry triangles whose all 3 vertices land exactly on the periodic domain will be allowed to participate in the simulation. This could result in incorrect particle contacts with these triangles which, especially for simulations with surface wear modification (see Enable and View Surface Wear Modification on an Imported Geometry (p. 379)), could result in inaccurate wear patterns. Enabling this setting might therefore be particularly important when simulating liners and lifters within SAG or HPGR mill slices.	Turns on or off

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

Figure 3.29: Solver | Output Properties



Output Properties		Description	Range
Properties	Particles	Select all particle properties that can be used in the post-processing	Turns on or off
	SPH	Select all SPH properties that can be used in the post-processing	Turns on or off
	Geometries	Select all Geometries properties that can be used in the post-processing	Turns on or off
	CFD	Select all CFD properties that can be used in the post-processing	Turns on or off
	Joints	Select all Joints properties that can be used in the post-processing	Turns on or off
	Contacts	Select all Contacts properties that can be used in the post-processing	Turns on or off
Module Properties	Boundary Collision Statistics	Select all Boundary Collision Statistics properties that can be used in the post-processing	Turns on or off
	Intra-particle Collision Statistics	Select all Intra-particle Collision Statistics properties that can be used in the post-processing	Turns on or off

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

Output Properties		Description	Range
	Joints Statistics	Select all Joints Statistics properties that can be used in the post-processing	Turns on or off
	Particle Instantaneous Energies	Select all Particle Instantaneous Energies properties that can be used in the post-processing	Turns on or off
	SPH Boundary Interaction Statistics	Select all SPH Boundary Interaction Statistics properties that can be used in the post-processing	Turns on or off
	SPH HTC Calculator	Select all SPH HTC Calculator properties that can be used in the post-processing	Turns on or off
	SPH-DEM Interaction Statistics	Select all SPH-DEM Interaction Statistics properties that can be used in the post-processing	Turns on or off

- Learn more About Contacts (p. 746)
- Learn more About Setting Global Preferences (p. 227)

See Also:

- I cannot find the Output Frequency parameter (p. 1336)
- I cannot find the Loading N-Steps parameter (p. 1336)
- I cannot find Particle Collision Statistics (p. 1330)
- About Study Parameters (p. 415)
- About Physics Parameters (p. 417)
- About Modules Parameters (p. 431)
- About Domain Settings Parameters (p. 452)
- Set Simulation Parameters (p. 414)

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

3.2.8. Geometries Parameters

Geometries are the physical boundaries that make up the components that are going to be simulated in Rocky. They can be default geometries which are included within Rocky and also geometries that you import from various CAD programs, or from Ansys Fluent or Ansys Motion setup files.

You can add as many individual geometry components as you want to your simulation in any combination you desire.

However, if you are using a particle or fluid inlet to release DEM particles or SPH elements into your simulation (see also About Adding and Editing Inlets and Outlets (p. 708)), before you are able to process the simulation, you must have a minimum of at least one inlet set up. If you are using a Volumetric Inlet, a simulation can be set up with no geometries in it.

Once you add the geometries you want, you can then change the parameters to achieve the behavior you want. The parameters you change can include the size, shape, and behavior of the default geometries included within Rocky, or special movements of the geometries you have imported, such as gates that lift or turn, for example.

At any point, you can see how your geometries look in a 3D View (see also Create and Modify a 3D View (p. 1040)).

What would you like to do?

- Add Geometry Components (p. 480)
- Import Wall or Surface Geometries (p. 487)

See Also:

- Set Simulation Parameters (p. 414)

ADD GEOMETRY COMPONENTS

There are two categories of geometries that you can add to your simulation: surfaces and walls; additionally Rocky offers you the option to use templates that can combine both types of geometries.

Tip:

Surfaces have a Normal Direction property, which indicates the direction of injection and removal for inlets and outlets and, also, the sign of flow measurements. In Rocky, there is an Invert Normal option, which inverts the normal o the surface and, as a consequence, inverts the directions for inlets/outlets and the flow measurements.

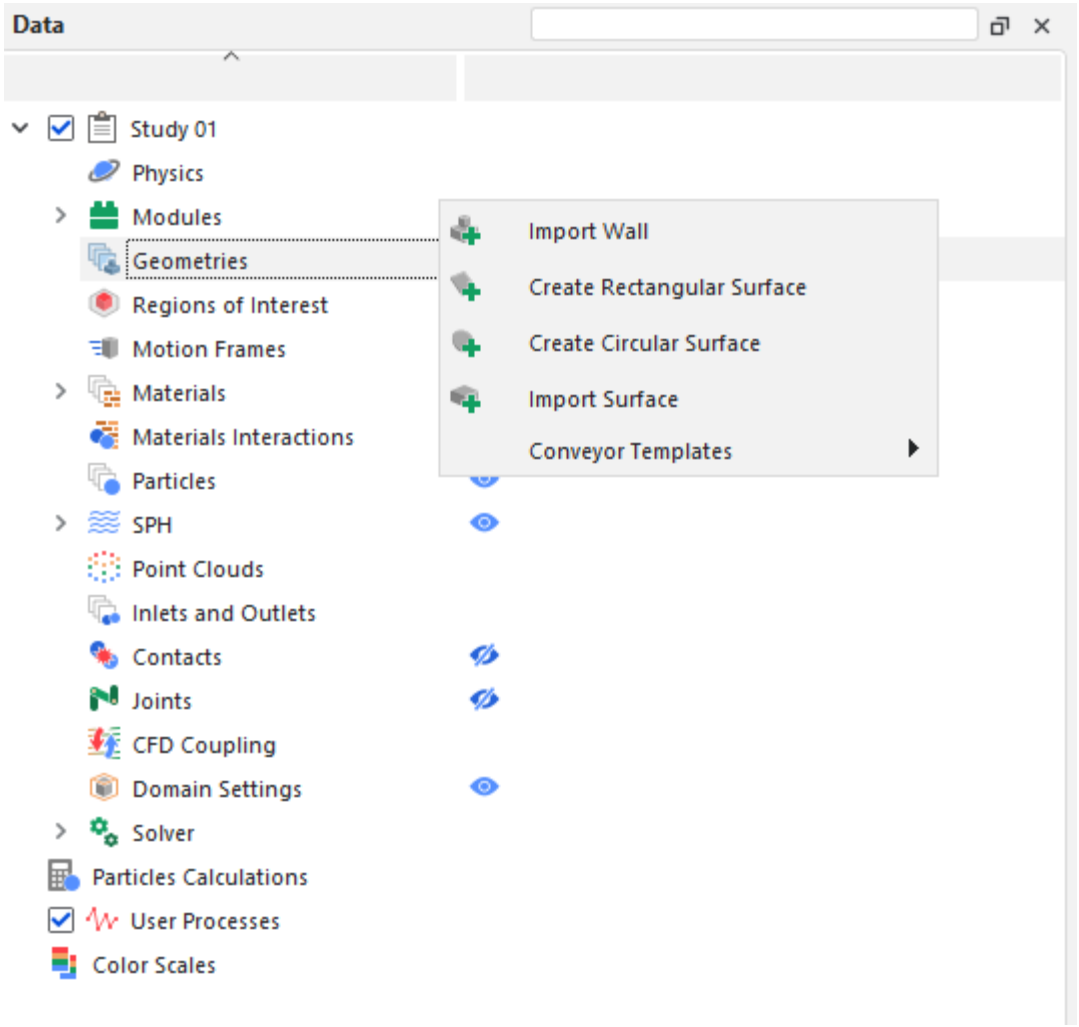
Surfaces are geometries that allow for the passage of SPH elements or DEM particles. They can be set up as inlets, outlets, or as flow measuring surfaces. They can also delimit regions in space that can be filled with fluid or particles. If you want a geometry to act as an barrier that can stop fluid or particles from flowing through it, this geometry must be set up as a wall.

IMPORTANT: If you want to use a surface as an Inlet or an Outlet, the surface must be planar (i.e. it should be contained into a single plane).

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

In Rocky geometries can be created or imported from external CAD/CAE softwares. The following options to create or import geometries are available (through the **Geometry** option on the Data panel) (see the image below):

Figure 3.30: Geometry options available in Rocky.



- **Import Wall**: allows you to import a geometry (see also Import Wall or Surface Geometries (p. 487)) that will act as a **Wall**.
- **Create Rectangular Surface**: allows you to create a rectangular geometry inside Rocky (see also Add New Circular and Rectangular Surfaces (p. 483)) that will act as a **Surface**.
- **Create Circular Surface**: allows you to create a circular geometry inside Rocky (see also Add New Circular and Rectangular Surfaces (p. 483)) that will act as a **Surface**.
- **Import Surface**: allows you to import a geometry (see also Import Wall or Surface Geometries (p. 487)) that will act as a **Surface**.
- **Conveyor Templates**: allows you to create templates of feeding or receiving conveyors (see also Add a Conveyor Template (p. 507)). These templates contain all the necessary **Walls** and **Surfaces**.

Conversion of Walls and Surfaces

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

In addition to the possibility to import Walls or Surfaces into your project, you can also convert Walls into Surfaces and Surfaces into Walls.

To do so, right-click the choose geometry, in the **Data Panel**, and then go to the option **Convert to**, as illustrated in the Figures below:

Figure 3.31: Convert Wall into Surface.

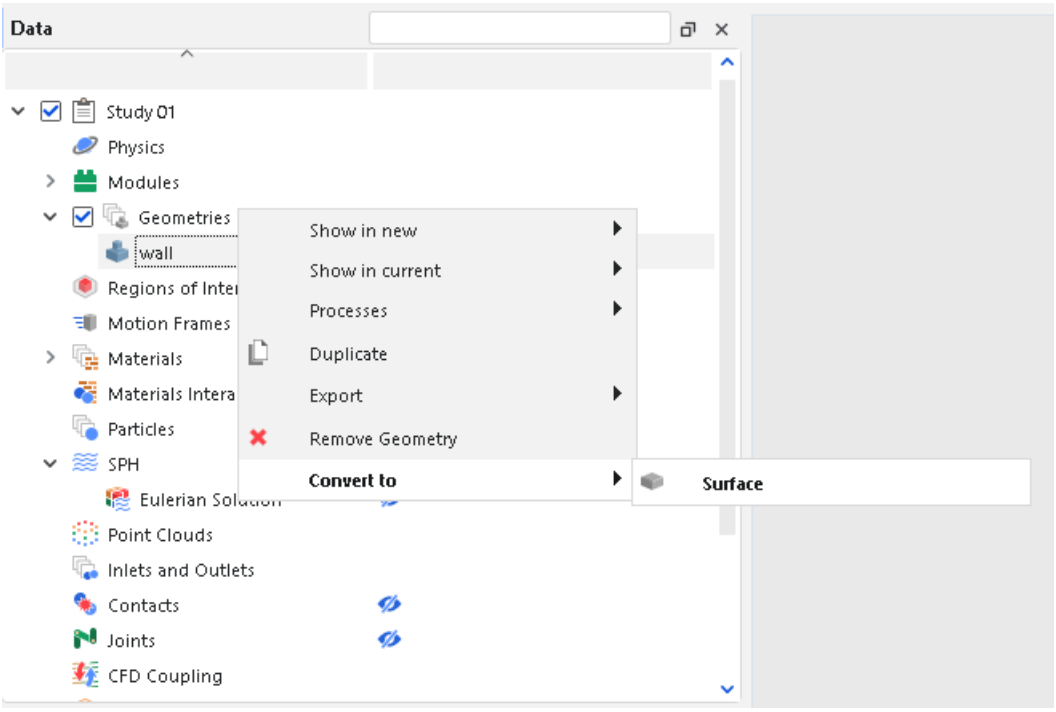
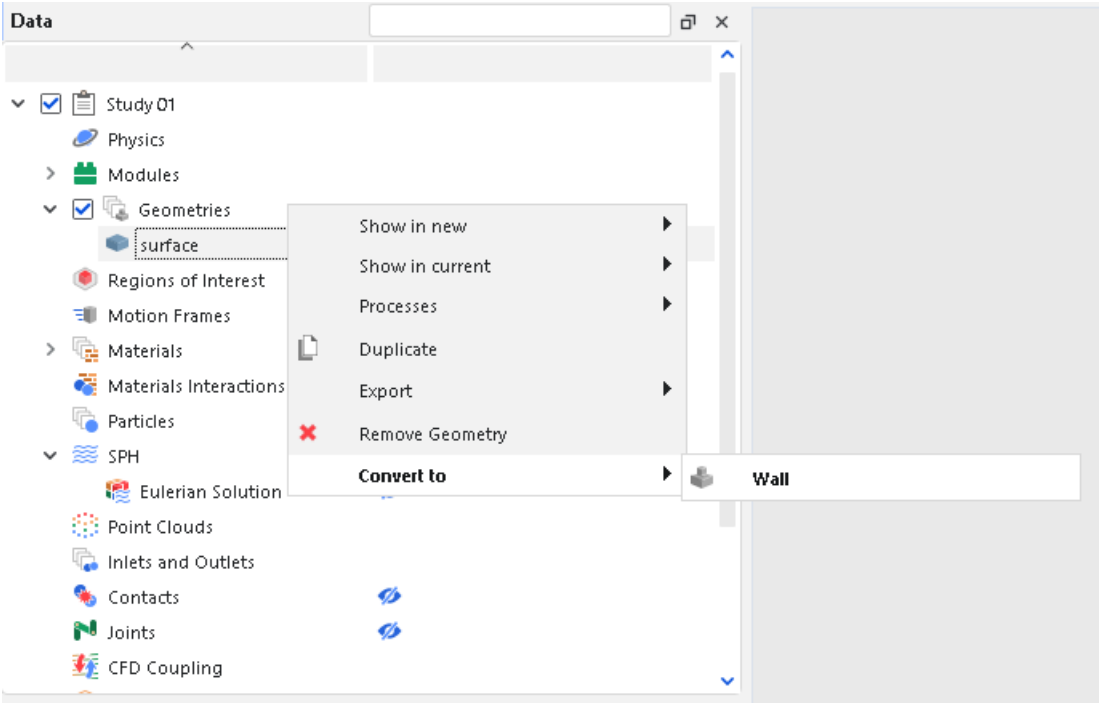


Figure 3.32: Convert Surface into Wall.



Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

What would you like to do?

- Learn more Import Wall or Surface Geometries (p. 487)
- Learn more Add New Circular and Rectangular Surfaces (p. 483)
- Add a Conveyor Template (p. 507)

ADD NEW CIRCULAR AND RECTANGULAR SURFACES

You can create rectangular and circular planar surfaces inside Rocky, eliminating the need of importing these geometries from an external CAD software. Since these geometries will act as an surface, you can use them to set inlets, outlets (see also Add and Edit Inlets and Outlets (p. 708)) and to perform flow measurements.

To create these surfaces, do all of the following:

1. From the **Data** panel, click **Geometries**, and then from the **Data Editors** panel, click the **Create Rectangular Surface** or **Create Circular Surface** button. A new Rectangular/Circular Surface component appears under **Geometries** in the **Data** panel. Tip: You may also access this functionality from the right-click menu on the **Data** panel.
2. From the **Data** panel, click the Rectangular/Circular component you just added and then from the **Data Editors** panel, on the main **Rectangular/Circular Surface** tab, enter the **Name** you want. You can also edit the properties listed in the table below.

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

Figure 3.33: Options available when creating a rectangular or circular surface

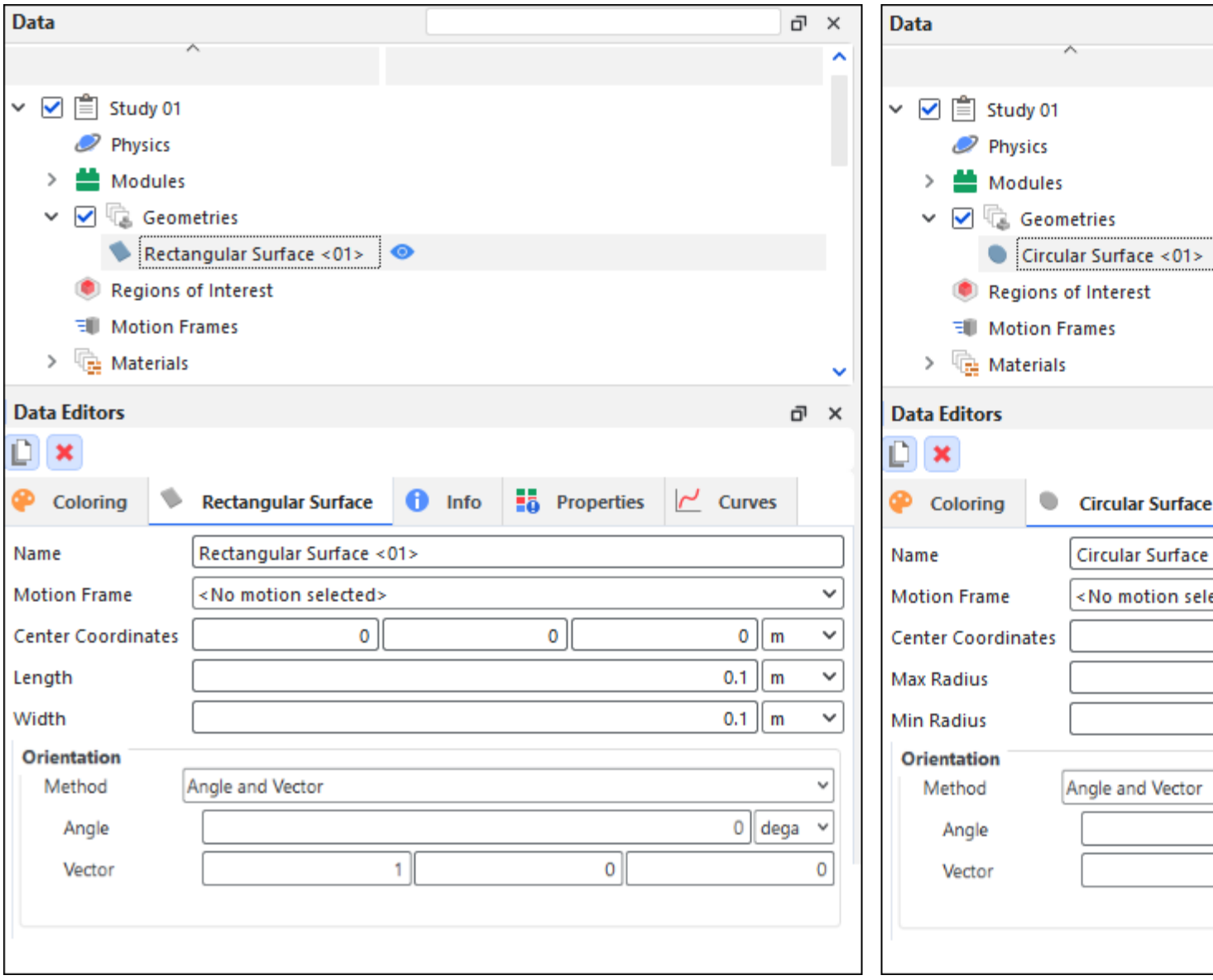


Table 1: Options available when creating a rectangular or circular surface

Setting	Description	Range
Name	Enables you to specify a unique identifier for the geometry component	99 character limit
Motion Frame	Enables you to select which Motion Frame you want assigned to the geometry. (See also About Creating and Applying Motion Frames (p. 533).)	Automatically provided
Center Coordinates	Enables you to define X, Y and Z coordinates of the center of the geometry.	Any value
Max Radius (Circular Surface Only)	Defines the radius of external limit of the circular geometry.	Positive Values

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

Setting	Description	Range
Min Radius (Circular Surface Only)	Defines the radius of internal limit of the circular geometry.	Positive Values
Lenght (Rectangular Surface Only)	Defines the transverse dimension of the rectangular geometry.	Positive Values
Width (Rectangular Surface Only)	Defines the longitudinal dimension of the rectangular geometry.	Positive Values
Method	Enables you to select how you want to define the orientation of the geometry. Specifically: • **Angles** enables you to define angles of rotation in three directions, the order of which you can also specify. • **Angle and Vector** enables you define one vector and one angle of rotation around it. • **Basis Vectors** enables you to define the X, Y, and Z directions of the geometry local basis.	Angles; Angle and Vector; Basis Vectors
Method Angles		
Order	When **Angles** is selected for **Orientation**, this defines the order in which the three **Rotation** fields will be applied.	XYZ; ZXY; YXZ; YZX; ZXY; ZYX
Local Angles	When **Angles** is selected for **Orientation**, this defines what coordinate system will be used as a basis for the angle specified. Specifically: • When enabled, the angle will be based on the local coordinate system. • When cleared, the angle will be based on the global coordinate system.	Turns on or off
Rotation	When **Angles** is selected for **Orientation**, this is the degree of cube rotation in each of the three directions specified by the **Order** provided.	Any value
Method Angle and Vector		
Angle	When **Angle and Vector** is selected for **Orientation**, this is the angle the cube will rotate around the **Vector** defined.	Any value
Vector	When **Angle and Vector** is selected for **Orientation**, this is the X, Y, and Z components that define the	No limit but the values entered

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

Setting	Description	Range
	vector around which the cube will rotate, using the **Angle** defined.	will be normalized
Method Basis Vectors		
X direction	When **Basis Vectors** is selected for **Orientation**, this is the coordinate values defining the first of three directional vectors that together define the final orientation of the cube. **Tip:** To ensure correct results, make sure you define this vector as orthogonal (perpendicular) to the other two vectors.	No limit but the values entered will be normalized
Y direction	When **Basis Vectors** is selected for **Orientation**, this is the coordinate values defining the second of three directional vectors that together define the final orientation of the cube. **Tip:** To ensure correct results, make sure you define this vector as orthogonal (perpendicular) to the other two vectors.	No limit but the values entered will be normalized
Z direction	When **Basis Vectors** is selected for **Orientation**, this is the coordinate values defining the third of three directional vectors that together define the final orientation of the cube. **Tip:** To ensure correct results, make sure you define this vector as orthogonal (perpendicular) to the other two vectors.	No limit but the values entered will be normalized

What would you like to do?

- Learn more Add a Conveyor Template (p. 507)
- Learn more About Adding and Editing Inlets and Outlets (p. 708)

See Also:

- Import Wall or Surface Geometries (p. 487)

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

- Replace an Imported Geometry File (p. 521)

IMPORT WALL OR SURFACE GEOMETRIES

All geometries that are not rectangular or circular, or have a template inside Rocky (such as conveyors), need to be imported. Imported geometries can be set up as both wall and surface. When imported as surfaces, the geometry can be used for flow measurements, as an inlet or an outlet (if planar). Importing geometry as walls allows for them to be used for additional calculations (such as surface wear or heat conduction).

Rocky supports importing geometries with the following file extensions: STL, DXF, XGL, CAS, CAS.GZ, CAS.H5, MSH, FMU, or DFG.

To import these surfaces, do all of the following:

1. From the **Data** panel, click **Geometries**, and then from the **Data Editors** panel, click the **Import Wall** or **Import Surface** button. Tip:You may also access this functionality from the right-click menu on the **Data** panel.

2. Select the file that you want to import into Rocky and click **Open**. Tip:To save time on larger projects, you may also want to import multiple components at once by multi-selecting several files on the **Select file to import** dialog.

3. Select the file that you want to import into Rocky and click **Open**

After choosing which file(s) you want to import, you are asked to define several options that enable you to determine the scale and unit of the imported file(s) (Figure 1). In addition, you can either choose to import the component names as they are by leaving the **Prefix for imported Geometries** blank, or you can add custom text to prefix the component name with whatever you like.

Imported geometry components may be replaced with a different geometry file prior to processing. (See also Replace an Imported Geometry File (p. 521).) Note: If you want to use a geometry for flow measurements or other post-processing analysis, it must be imported into Rocky **before** the simulation has started. In post-processing Rocky is not able to import a geometry without needing to restart the simulation.

Import File Info Parameters

Use the figure and table below to understand the various import parameters you can set for your imported geometries and then use the procedures that follow to learn about how to add geometry components to your simulation setup.

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

Figure 3.34: Import File Info dialog

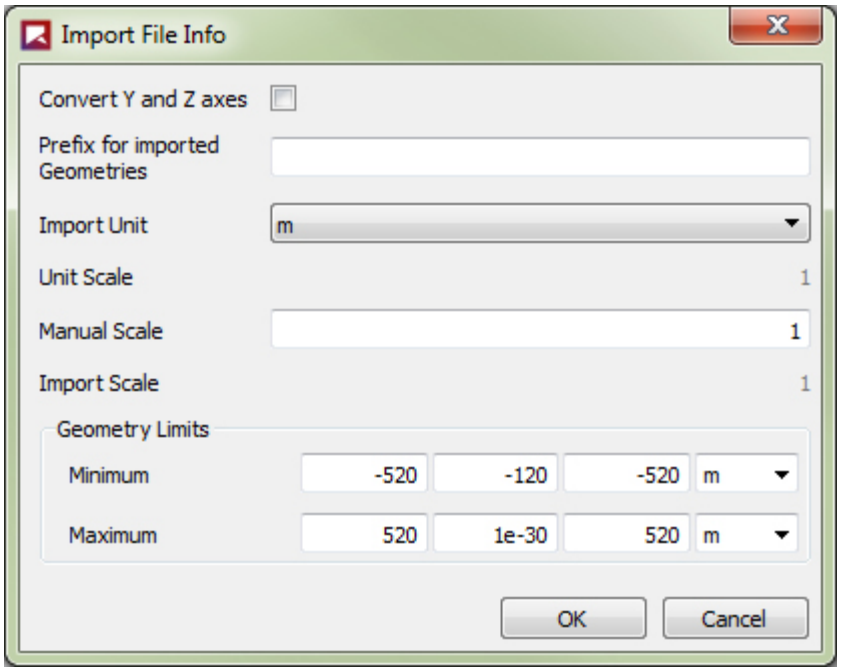


Table 1: Import options displayed in the Rocky dialog

Setting	Description	Range
Convert Y and Z axes	Selecting this option will change the axis of the imported geometry.	Turns on or off
Prefix for imported Geometries	When cleared (blank), the name of the imported file will be displayed in the Geometries list. When defined, this additional name will be displayed directly before the imported name in the Geometries list. If the geometry contains multiple components, each component will have this same prefix followed by the (unique) imported name of the file.	99 character limit
Import Unit	Enables you to change the units of the imported geometry.	Various units of length
Unit Scale	Displays the unit scale based upon the **Import Unit** set. For example, if **Import Unit** is left as the default value, the Unit Scale will be 1.	Automatically determined
Manual Scale	Enables you to manually adjust the scale by any factor you want. Leaving the value at 1 will have no additional affect upon the scale.	All values
Import Scale	Displays the final import scale based upon the **Manual Scale** and **Import Unit** values set. For example, if both those options are left as the default values, the Import Scale will be 1.	Automatically determined
Geometry Limits		

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

Setting	Description	Range
Minimum	The coordinates (in X Y Z format) of the lowest points geometry triangles are drawn.	No limit
Maximum	The coordinates (in X Y Z format) of the highest points geometry triangles are drawn.	No limit

Important:

If an STL file is loaded and a triangle is missing, it might be because the triangle has an area equal to zero (two or three vertices at the same position). Rocky automatically deletes triangles with this characteristic to avoid issues with the simulation. If this occurs, check the coordinates of the missing triangles and ensure that none are coincident.

Importing a Geometry as Surface

Use the figure and table below to understand the various import parameters you can set for a geometry that was imported as a surface.

Prior to processing, Geometry components may be replaced with a different geometry file, (see also Replace an Imported Geometry File (p. 521)), and have a Motion Frame assigned to them (see also Apply a Motion Frame to an Imported Geometry (p. 569).)

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

Figure 3.35: Options available for geometries imported as surfaces

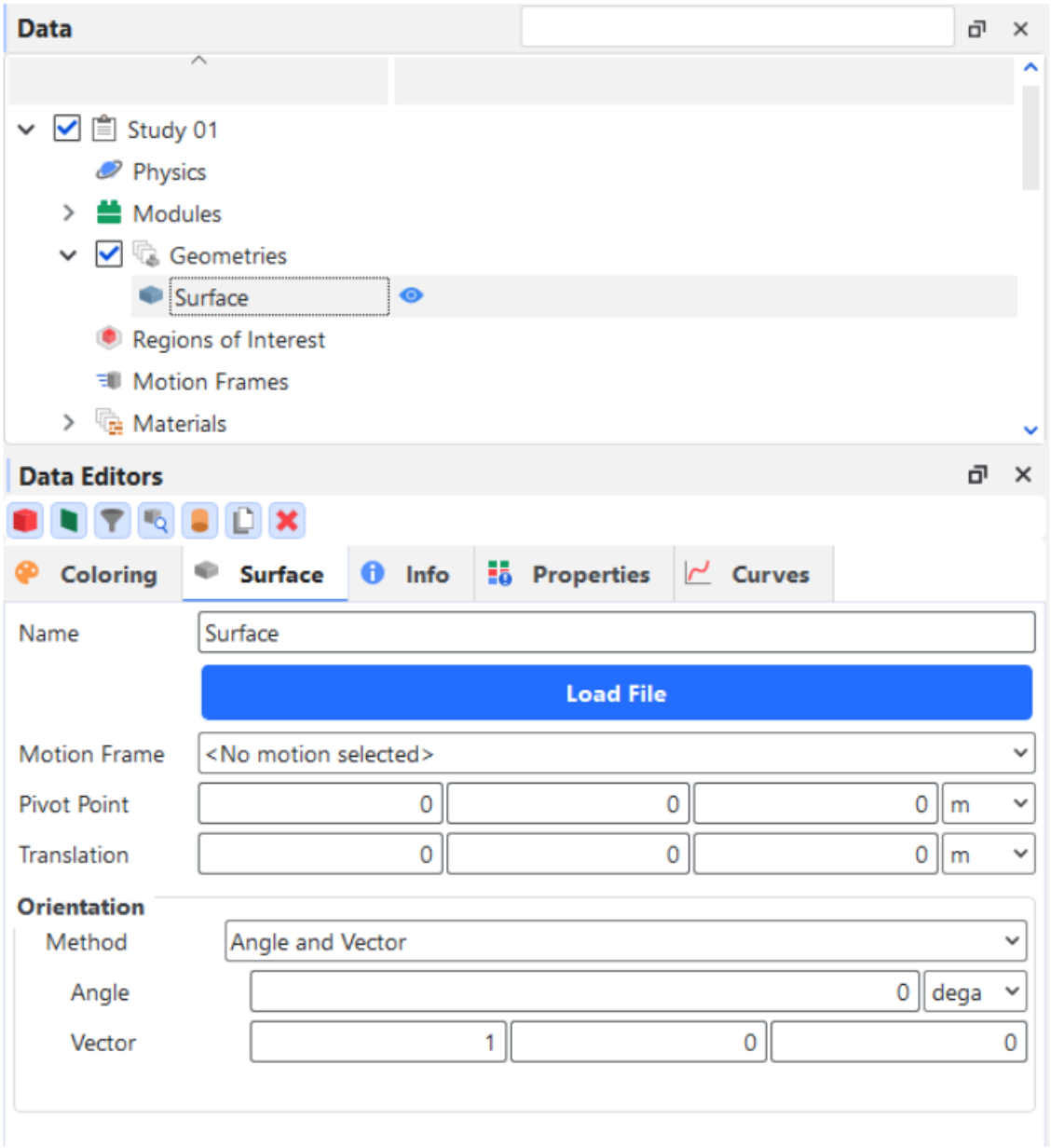


Table 2: Options available for geometries imported as surfaces

Setting	Description	Range
Name	Enables you to specify a unique identifier for the geometry component	99 character limit
Motion Frame	Enables you to select which Motion Frame you want assigned to the geometry. (See also About Creating and Applying Motion Frames (p. 533).)	Automatically provided

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

Setting	Description	Range
Pivot Point	The coordinate location of the point around which the surface will pivot, as specified in the X, Y, and Z directions. **Note:** In the calculation order, the rotation is calculated before the translation. In this way, the Pivot Point presented in the UI does not include the translation values, therefore to obtain the real value, it is necessary to sum the pivot point values with the translation values.	No limit
Translation	Enables you to move the geometry in the X, Y and Z directions	Zero or Positive Values
Orientation	Enables you to select how you want to define the orientation of the geometry. Specifically: • **Angles** enables you to define angles of rotation in three directions, the order of which you can also specify. • **Angle and Vector** enables you define one vector and one angle of rotation around it. • **Basis Vectors** enables you to define the X, Y, and Z directions of the geometry local basis.	Angles; Angle and Vector; Basis Vectors
Orientation Angles		
Order	When **Angles** is selected for **Orientation**, this defines the order in which the three **Rotation** text fields will be applied.	XYZ; ZXY; YXZ; YZX; ZXY; ZYX
Local Angles	When **Angles** is selected for **Orientation**, this defines what coordinate system will be used as a basis for the angle specified. Specifically: • When enabled, the angle will be based on the local coordinate system. • When cleared, the angle will be based on the global coordinate system.	Turns on or off
Rotation	When **Angles** is selected for **Orientation**, this is the degree of cube rotation in each of the three directions specified by the **Order** provided.	Any value

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

Setting	Description	Range
Orientation Angle and Vector		
Angle	When **Angle and Vector** is selected for **Orientation**, this is the angle the cube will rotate around the **Vector** defined.	Any value
Vector	When **Angle and Vector** is selected for **Orientation**, this is the X, Y, and Z components that define the vector around which the cube will rotate, using the **Angle** defined.	No limit but the values entered will be normalized
Orientation Basis Vectors		
X direction	When **Basis Vectors** is selected for **Orientation**, this is the coordinate values defining the first of three directional vectors that together define the final orientation of the cube. **Tip:** To ensure correct results, make sure you define this vector as orthogonal (perpendicular) to the other two vectors.	No limit but the values entered will be normalized
Y direction	When **Basis Vectors** is selected for **Orientation**, this is the coordinate values defining the second of three directional vectors that together define the final orientation of the cube. **Tip:** To ensure correct results, make sure you define this vector as orthogonal (perpendicular) to the other two vectors.	No limit but the values entered will be normalized
Z direction	When **Basis Vectors** is selected for **Orientation**, this is the coordinate values defining the third of three directional vectors that together define the final orientation of the cube. **Tip:** To ensure correct results, make sure you define this vector as orthogonal (perpendicular) to the other two vectors.	No limit but the values entered will be normalized

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

Importing a Geometry as Wall

After you have imported a geometry as a wall, you can then edit the Geometry, Mass, Wear, and Replication parameters that define how the geometry behaves in the simulation.

Prior to processing, Geometry components may be replaced with a different geometry file, (see also Replace an Imported Geometry File (p. 521)), and have a Motion Frame assigned to them (see also Apply a Motion Frame to an Imported Geometry (p. 569).)

After processing, Geometry components can be exported out of Rocky into an STL file, which is especially useful after the geometry surface has been modified by wear. (See also Export a Geometry Component to an STL File (p. 520).)

ABOUT MASS PARAMETERS

The mass settings are only useful when a Motion Frame with either **Free Body Translation** or **Free Body Rotation** has been applied to the geometry. This is to ensure that the component behaves correctly in terms of gravity and any additional (prescribed) force/moments effects, as well as interactions with particles.

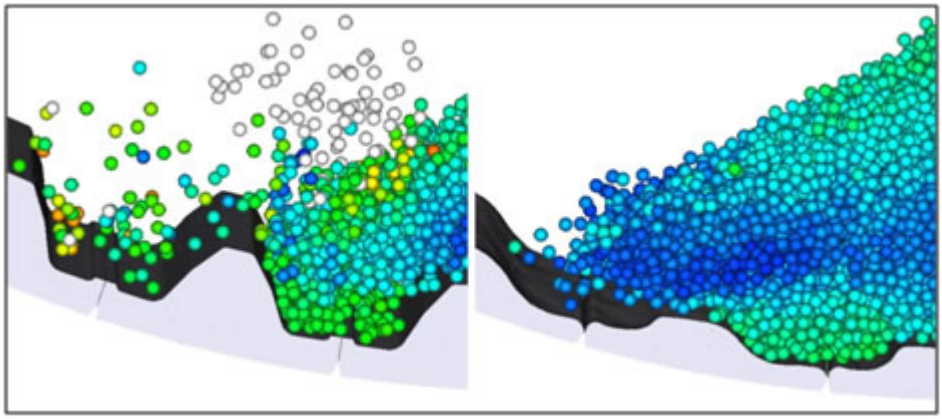
Tip:

The Principal Moment of Inertia values can be determined in your CAD program.

ABOUT WEAR PARAMETERS

The wear settings enable you to modify the surface of your geometry based upon interactions with particles. The purpose of this feature is to simulate how materials-such as equipment liners-might physically wear down over time (Figure 3). (See also Enable and View Surface Wear Modification on an Imported Geometry (p. 379).)

Figure 3.36: Example of a mill liner being modified by wear



Tips:

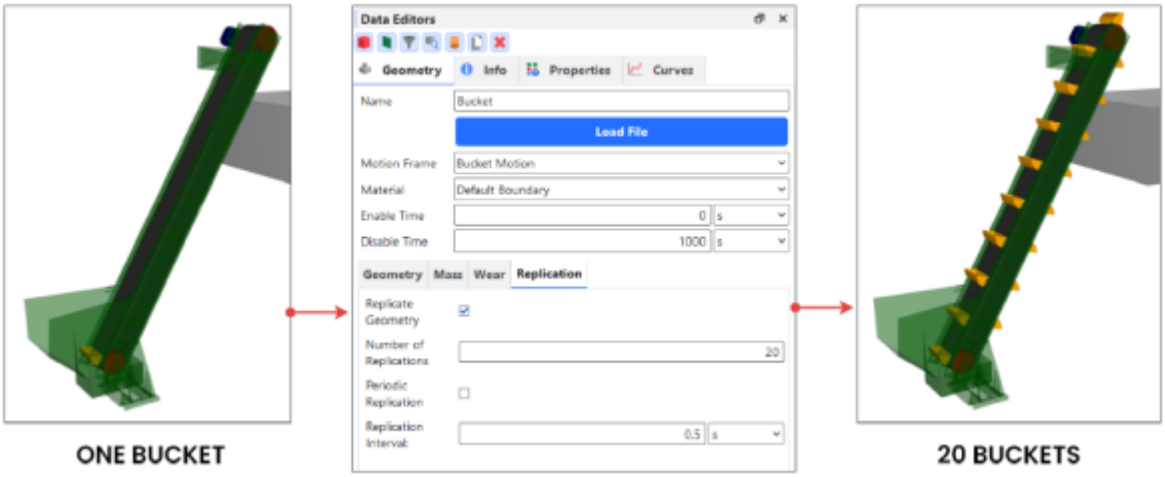
- To see a walk-through example of surface wear modification, refer to the following Workshops:
- Tutorial - SAG Mill in the *Rocky Tutorial Guide*
- Tutorial - High Pressure Grinding Roll (HPGR) in the *Rocky Tutorial Guide*

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

ABOUT REPLICATION PARAMETERS

The replication settings, when coupled with Motion Frames and periodic motions, enable you to reproduce a geometry component at intervals along the path of the assigned motion. This can be useful for turning a single bucket into a full bucket elevator, for example (Figure 4).

Figure 3.37: Replication example showing how one bucket in a bucket elevator can be turned into 20



In this version of Rocky, replication can also be previewed in a Motion Preview window before processing the simulation. (See also Preview a Motion in 3D (p. 573).)

Note:

Geometry replication features are not compatible with free body motions. (See also About Creating and Applying Motion Frames (p. 533).)

Tip:

To see a walk-through example of replicating your geometries using periodic motions, refer to the following Workshop: Tutorial - **Bucket Conveyor** in the *Rocky Tutorial Guide*.

ABOUT MODULES PARAMETERS

If you have enabled an external Module (p. 1391) that affects your Geometries settings, you might also have a separate **Modules** sub-tab with additional settings that you can define (Figure 9). Refer to the Module's documentation (if provided) for more information. (See also Rocky Simulation Entities that can be Affected by Modules (p. 223).)

Imported Wall Parameters

Use the following figures and table to understand the various Geometry, Mass, Wear, and Replication parameters that you can set for an imported geometry.

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

Figure 3.38: Imported wall, Geometry parameters in the Data Editors panel

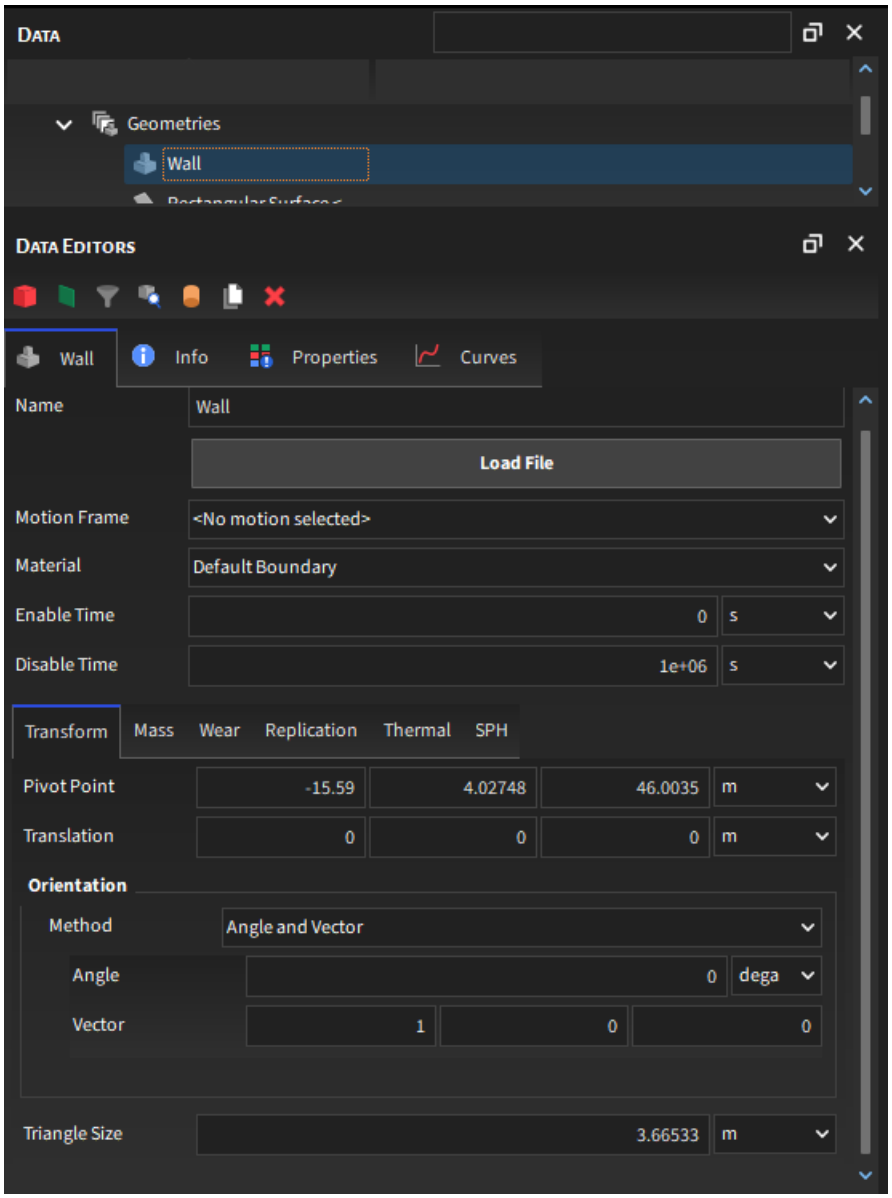
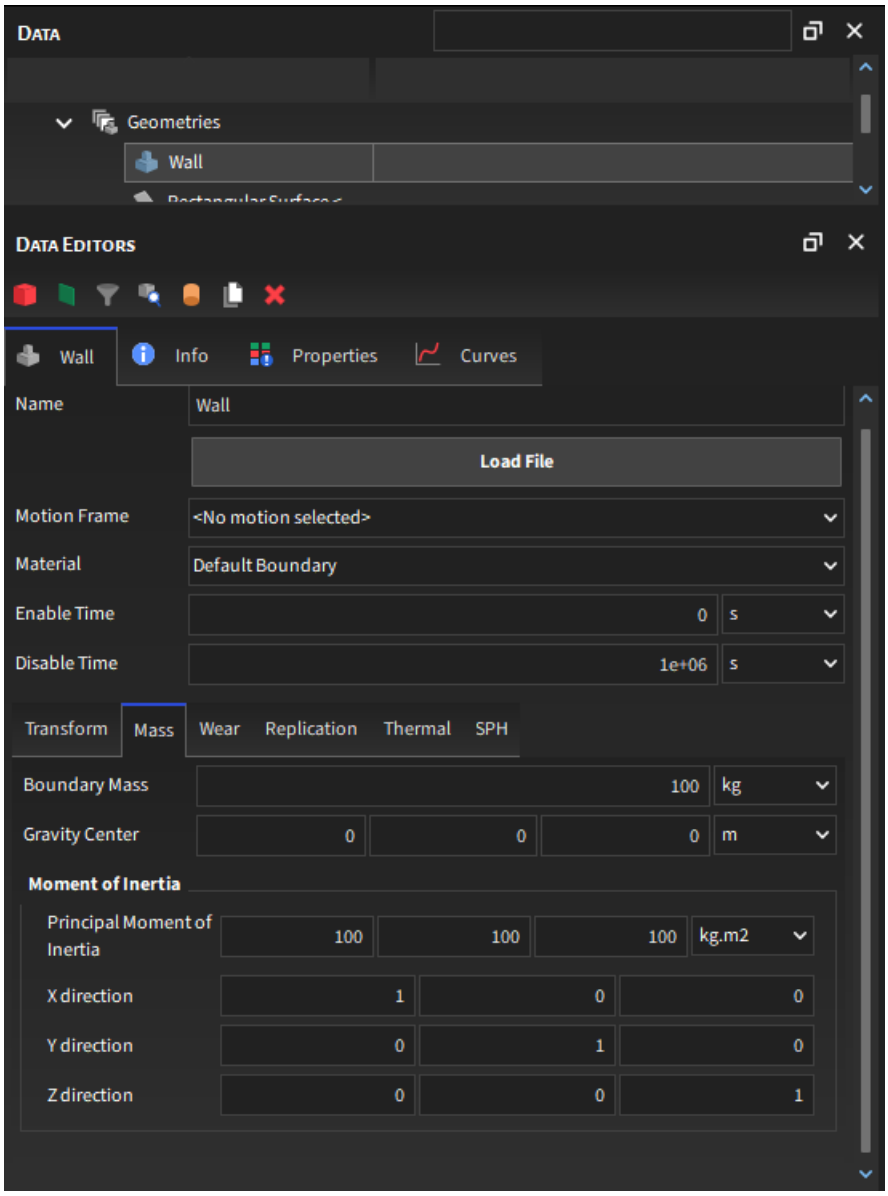


Figure 3.39: Imported wall, Mass parameters in the Data Editors panel



Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

Figure 3.40: Imported wall, Wear parameters in the Data Editors panel

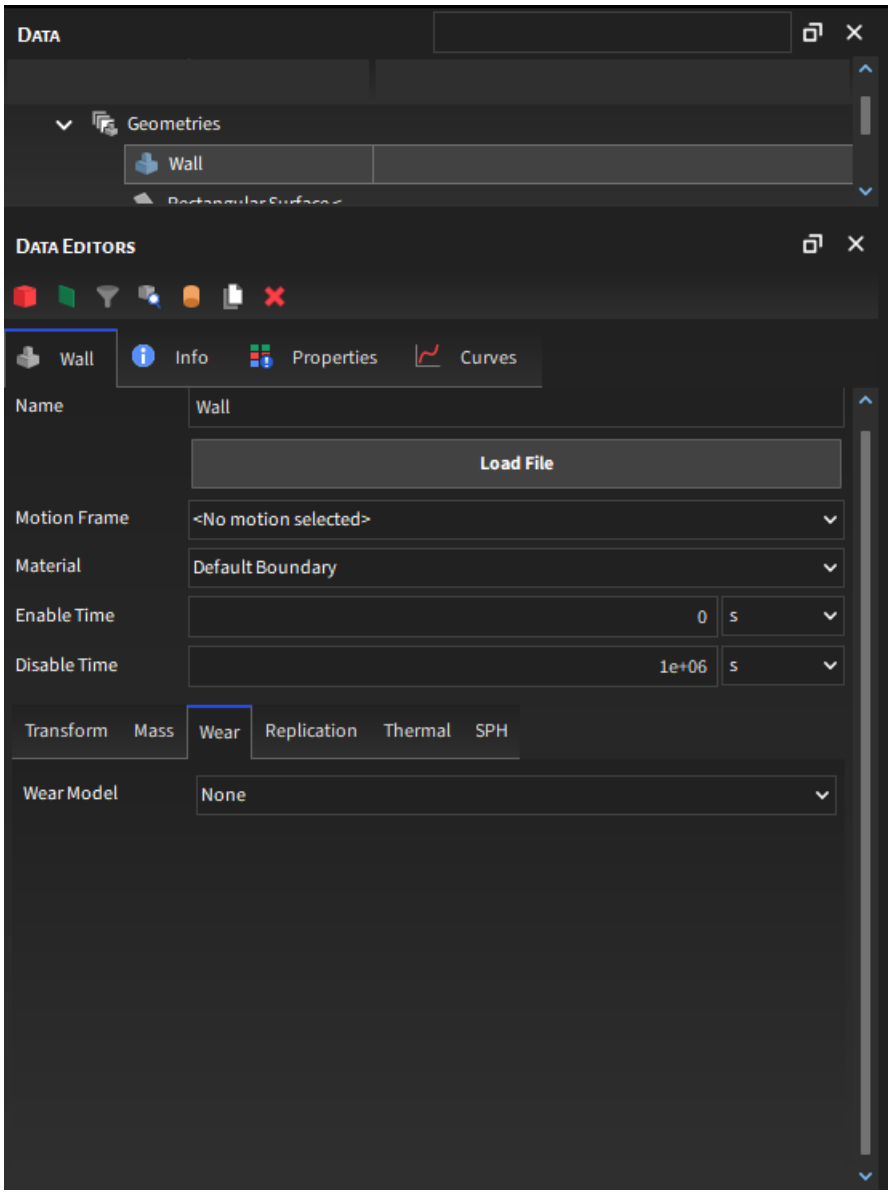
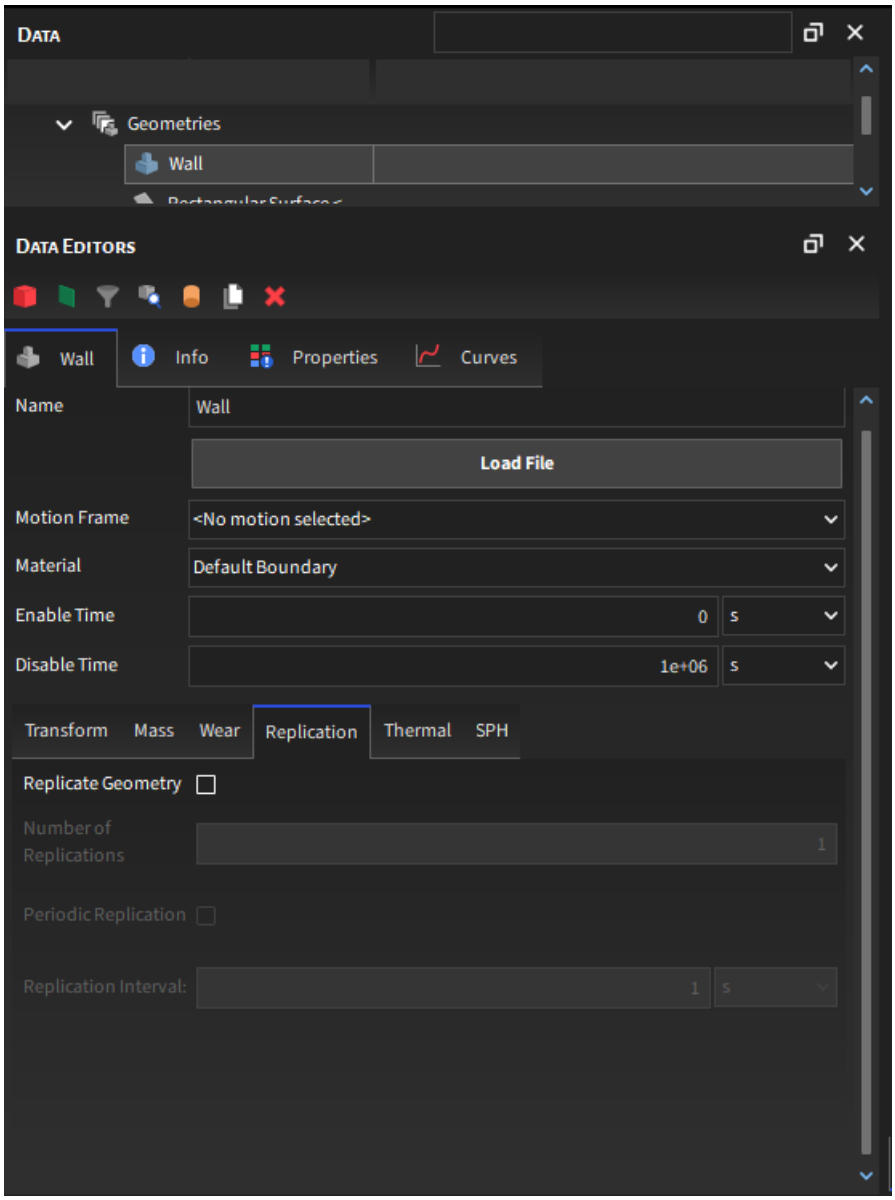
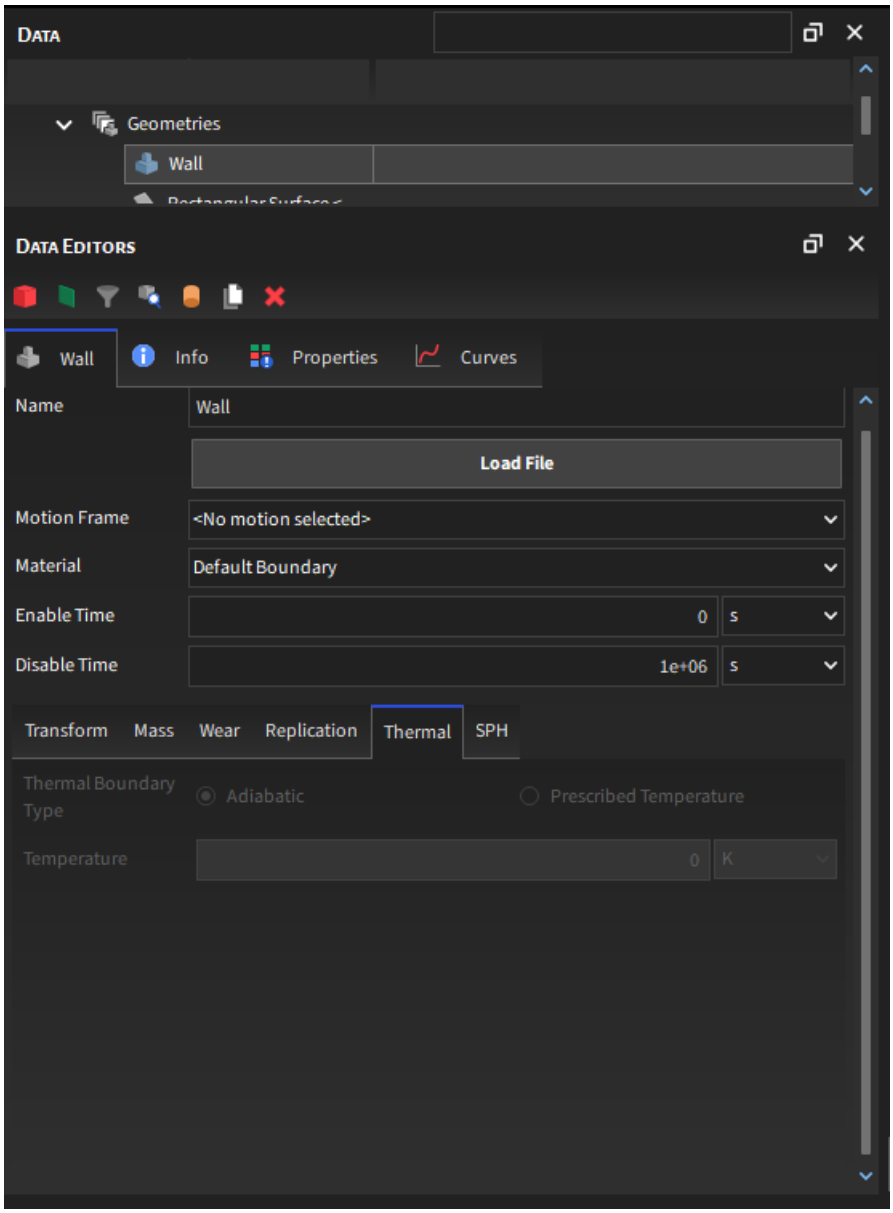


Figure 3.41: Imported wall, Replication parameters in the Data Editors panel



Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

Figure 3.42: Imported wall, Thermal parameters in the Data Editors panel



Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

Figure 3.43: Imported wall, SPH parameters in the Data Editors panel

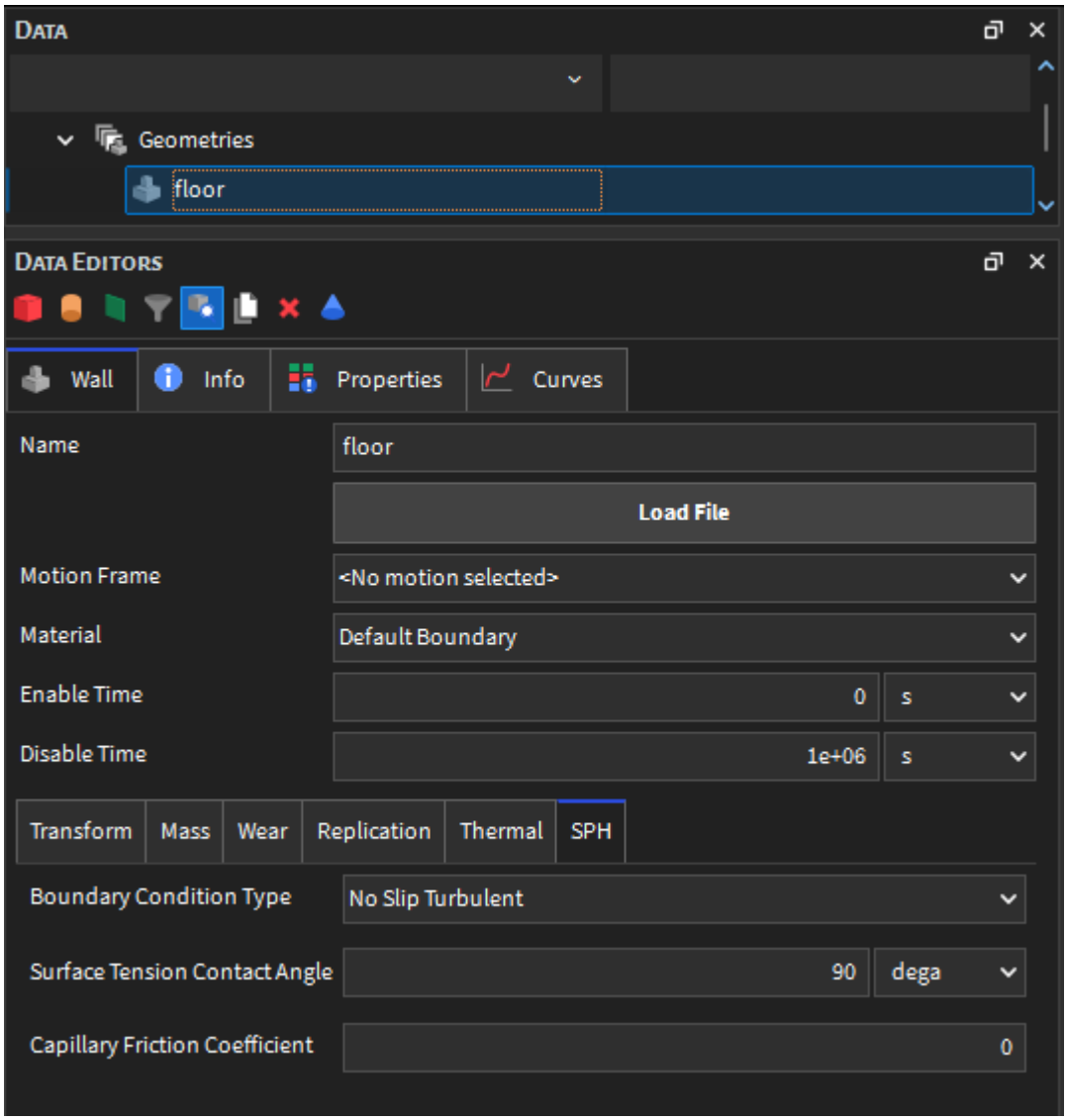


Figure 3.44: Imported wall, Modules tab in the Data Editors panel

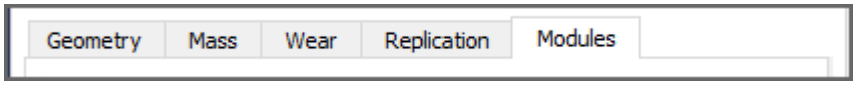


Table 3 : Imported wall parameter options (Geometry, Mass, Wear, and Replication subtabs)

Setting	Description	Range
Name	Enables you to specify a unique identifier for the wall component.	No limit
Motion Frame	Enables you to select which Motion Frame you want assigned to the geometry. (See also About Creating and Applying Motion Frames (p. 533).)	Automatically provided

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

Setting	Description	Range
Material	Defines the density and Young's Modulus of the wall component based upon the options you have set in the Materials list. (See About Modifying Material Compositions (p. 591) for more information.) When **Thermal Model** is enabled (see also About Physics Parameters (p. 417)), this also defines the specific heat and Poisson's ratio values, both of which are set in the Materials list.	List is based upon the Materials that have been defined
Enable Time	The time you want the geometry to start interacting with particles during the simulation. **Tip:** To have the geometry interact with particles at simulation onset, keep **Enable Time** as zero (0). Note: *If you choose to use parametric expressions in this field, know that only the resulting value and not the variables and/or mathematical functions you enter will be retained in any project copies you save for restart purposes.* (See also I get a "Links removed" message when I save my project for restart purposes (p. 1381).)	Positive values **Tip:** Check the Status panel to ensure that any variables or mathematical functions you might use results in valid values. (See also Double-Click the Status Panel to Jump to the Appropriate UI Location (p. 117).)
Disable Time	The time you want the geometry to stop interacting with particles during the simulation. **Tip:** To have the geometry interact with particles for the duration of the simulation, keep	Positive values **Tip:** Check the Status

Setting	Description	Range
	Disable Time as 1e+06 or the maximum length of your simulation. Note: *If you choose to use parametric expressions in this field, know that only the resulting value and not the variables and/or mathematical functions you enter will be retained in any project copies you save for restart purposes.* (See also I get a "Links removed" message when I save my project for restart purposes (p. 1381).)	panel to ensure that any variables or mathematical functions you might use results in valid values. (See also Double-Click the Status Panel to Jump to the Appropriate UI Location (p. 117).)
Transform		
Pivot Point	The coordinate location of the point around which the wall will pivot, as specified in the X, Y, and Z directions. **Note:** When Rocky calculates the transformation of coordinates of walls/surfaces, the operation of rotating the geometry around the pivot point comes first, then the translation operation is applied on those rotated coordinates. As a consequence, the pivot point that is inputted in the UI is the position of the pivot point before the translation, while the pivot point that appears in the 3D view is the pivot point after the translation operation.	No limit
Translation	Enables you to move the geometry in the X, Y and Z directions.	Zero or Positive Values
Orientation	Enables you to select how you want to define the orientation of the wall shape. Specifically:	Angles; Angle and Vector;

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

Setting	Description	Range
	• **Angles** enables you to define angles of rotation in three directions, the order of which you can also specify. • **Angle and Vector** enables you to define one vector and one angle of rotation around it. • **Basis Vectors** enables you to define the X, Y, and Z directions of the cube's local basis.	Basis Vectors.
Orientation: Angle and Vector		
Angle	When Angle and Vector is selected for Orientation, this is the angle the wall will rotate around the defined Vector.	Any value
Vector	When Angle and Vector is selected for Orientation, this is the X, Y, and Z components that define the vector around which the wall will rotate, using the defined Angle.	No limit but the values entered will be normalized
Orientation: Basis Vector		
X direction	When Basis Vectors is selected for Orientation, this is the coordinate values defining the first of three directional vectors that together define the final orientation of the wall. Tip: To ensure correct results, make sure you define this vector as orthogonal (perpendicular) to the other two vectors.	No limit but, the values entered will be normalized
Y direction	When Basis Vectors is selected for Orientation, this is the coordinate values defining the second of three directional vectors that together define the final orientation of the wall. Tip: To ensure correct results, make sure you define this vector as orthogonal (perpendicular) to the other two vectors.	No limit but,the values will be normalized
Z direction	When Basis Vectors is selected for Orientation, this is the coordinate values defining the third of three directional vectors that together define the final orientation of the wall. Tip: To ensure correct results, make sure you define this vector as orthogonal (perpendicular) to the other two vectors.	No limit but,the values will be normalized
Orientation: Angles		
Order	When Angles is selected for Orientation, this defines the order in which the three Rotation text fields will be applied.	XYZ; ZXY;

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

Setting	Description	Range
		YXZ; YZX; ZXY; ZYX.
Local Angles	When Angles is selected for Orientation, this defines what coordinate system will be used as a basis for the angle specified. Specifically: • When enabled, the angle will be based on the local coordinate system. • When cleared, the x angle will be based on the global coordinate system.	Turns on or off
Rotation	When Angles is selected for Orientation, this is the degree of wall rotation in each of the three directions specified by the Order provided.	Any value
Triangle Size	Size of the triangular components into which the boundary is divided. This value is used both for refining the boundary mesh (this will not happen if the imported Triangle Size is already finer than this value) and for averaging purposes during wear and surface wear modification calculations. Therefore, in general, the smaller the Triangle Size, the more accurate the wear calculations but the longer the processing time. (See also View a Color Map of Wear on the Default Belt or Imported Geometry Itself (p. 381) and View Surface Wear Modification on an Imported Geometry (p. 379).)	Positive values
Mass		
Boundary Mass	When **Free Body Translation** or **Free Body Rotation** are defined for the Motion Frame applied to this geometry, this is the mass of the geometry.	Positive values
Gravity Center	When **Free Body Translation** or **Free Body Rotation** are defined for the Motion Frame applied to this geometry, this is the location in coordinates of the center point of gravity for the geometry. This can only be visualized in a 3D view, and when t=0. **Note:** The Imported Wall's Gravity Center is automatically set equal to the Wall's	No limit

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

Setting	Description	Range
	Pivot Point. This value can be changed when setting up the simulation.	
Mass (Moments of Inertia)		
Principal Moment of Inertia	When **Free Body Translation** or **Free Body Rotation** are defined for the Motion Frame assigned to this geometry (see also About Creating and Applying Motion Frames (p. 533)), these are the principal moments of inertia along the X, Y, and Z axes defined below.	Positive values greater than but not equal to zero
X direction	The X, Y, and Z components that define the X axis for the **Principal Moment of Inertia**. Note: This is based upon the global coordinate system.	No limit but the values entered will be normalized, and the base must be positive orthonormal
Y direction	The X, Y, and Z components that define the Y axis for the **Principal Moment of Inertia**. Note: This is based upon the global coordinate system.	No limit but the values entered will be normalized, and the base must be positive orthonormal
Z direction	The X, Y, and Z components that define the Z axis for the **Principal Moment of Inertia**. Note: This is based upon the global coordinate system.	No limit but the values entered will be normalized, and the base must be positive orthonormal
Wear		
Wear Model	The calculation used to perform wear modifications of the geometry surface. Unless you have enabled a custom *external Modules* (p. 1391) that defines a different Wear model, the only option listed will be **Shear Work Proportionality (Archard's Law)**. This model works by ensuring that the volume of geometry removed is proportional to the shear wear applied to the geometry.	None; Shear Work Proportionality (Archard's Law) Note: If you have a Module that adds in an additional Wear law, you must use that law at least once in your simulation project if the Module is enabled. Refer to the Module's documentation (if

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

Setting	Description	Range
		provided) for more information. (See also Rocky Simulation Entities that can be Affected by Modules (p. 223).)
Volume/Shear Work Ratio	The amount of surface volume that is removed per shear work applied. **Note:** This is a calibration step that comes from real-world wear data you have collected using similar types of liners.	Positive values
Replication		
Replicate Geometry	Enables Rocky to replicate the geometry along the path of the Motion Frame that is applied to it. This is useful for creating several exact copies of a geometry along the same movement path, such as turning a single pan into a full apron feeder, a single bucket into a full bucket conveyor (Figure 4), and so on. Works best in conjunction with periodic motions. (See also About Creating and Applying Motion Frames (p. 533).) When used without periodic motions, the replicas will still start in different positions along the path of the Motion Frame, but will all eventually stop in the same final position (i.e., overlapping one another) when the motion frame's stop time is reached.	Turns on or off
Number of Replications	Sets the amount of times (copies) you want the geometry to be replicated.	Positive integer values
Periodic Replication	Determines how the time interval between geometry replications is set. Specifically, when: • **Enabled** (checked), this allows you to set the **Replication Period** value, which is the total amount of time during which the geometry replications will occur. • **Disabled** (unchecked), this allows you to set the Replication Interval, which sets the amount of time between geometry replications. These intervals may be useful for specifying gaps between buckets along a bucket conveyor, for example.	Turns on or off

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

Setting	Description	Range
Replication Period	When **Periodic Replication** is enabled (checked), this sets the total amount of time during which geometry replications will occur. The interval between individual replications can be determined by dividing this value by the Number of Replications that is set. For example, if you set this value to 3 and Number of Replications is also 3, then the geometry will replicate every 1 s.	Positive values
Replication Interval	When **Periodic Replication** is cleared (unchecked), this sets the amount of time between geometry replications. For example, if you set this value to 1, the geometry will be replicated every 1 s until the **Number of Replications** value has been achieved.	Positive values
Modules		
(Varies)	These settings are specific to only certain *external Modules* (p. 1391) and are therefore not documented in the Rocky User Manual. Refer to the Module's documentation (if provided) for more information. (See also Rocky Simulation Entities that can be Affected by Modules (p. 223).)	(Varies)

What would you like to do?

- Replace an Imported Geometry File (p. 521)
- Learn more Add Geometry Components (p. 480)
- Learn more About Creating and Applying Motion Frames (p. 533)

See Also:

- Edit the Parameters for a Geometry (p. 519)
- Set Simulation Parameters (p. 414)

ADD A CONVEYOR TEMPLATE

Conveyors template are included in Rocky, allowing you to use Feed and Receiving conveyors without the need of importing a custom geometry. One of the benefits of using a default Conveyor is that you can easily adjust the speed of the conveyor itself, and even set it up to accelerate from or decelerate to a full stop, which is particularly useful for testing plant start-up or shut-down scenarios.

One of the main drawbacks of using a Conveyor template is that while you can certainly analyze the wear on the belt using color mapping (see also View a Color Map of Wear on the Default Belt or Imported Geometry Itself (p. 381)), default geometries, including Feed Conveyors and Receiving Conveyors, do not support showing surface wear modification (see also View Surface Wear Modification on an Imported Geometry (p. 379)) on the boundaries. So if you are interested in doing that kind of wear analysis, you will want to import a custom feed conveyor instead (see also Import Wall or Surface Geometries (p. 487).).

Note: If you have enabled an external Module (p. 1391) that affects your Conveyor template settings, you might also have a separate **Modules** sub-tab with additional settings that you can define (Figure

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

6). Refer to the Module's documentation (if provided) for more information. (See also Rocky Simulation Entities that can be Affected by Modules (p. 223).)

To add a conveyor template to your simulation:

1. From the **Data** panel, right-click **Geometries**, and then do one of the following:

   - Point to **Conveyor Templates**, and then click either **Create Feed Conveyor** or **Create Receiving Conveyor**.

   The item you chose is now listed alphabetically (case sensitive) in the **Data** panel below **Geometries**.

2. Repeat step 1 for as many default geometries that you want in your simulation.

Adding a Feed Conveyor Template

Feed Conveyors come with a built-in inlet (p. 1392) (equal to the size, location, and orientation of the top of the **Feeder Box**), and can be adjusted to account for different pulley and skirtboard sizes, belt and roll configurations, and more.

Note: You cannot set a fluid inlet in a Feed Conveyor template.

Default Feed Conveyor Parameters

Use the figures and table below to understand the various parameters you can set for a default feed conveyor.

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

Figure 3.45: Default Feed Conveyor, Geometry parameters in the Data Editors panel

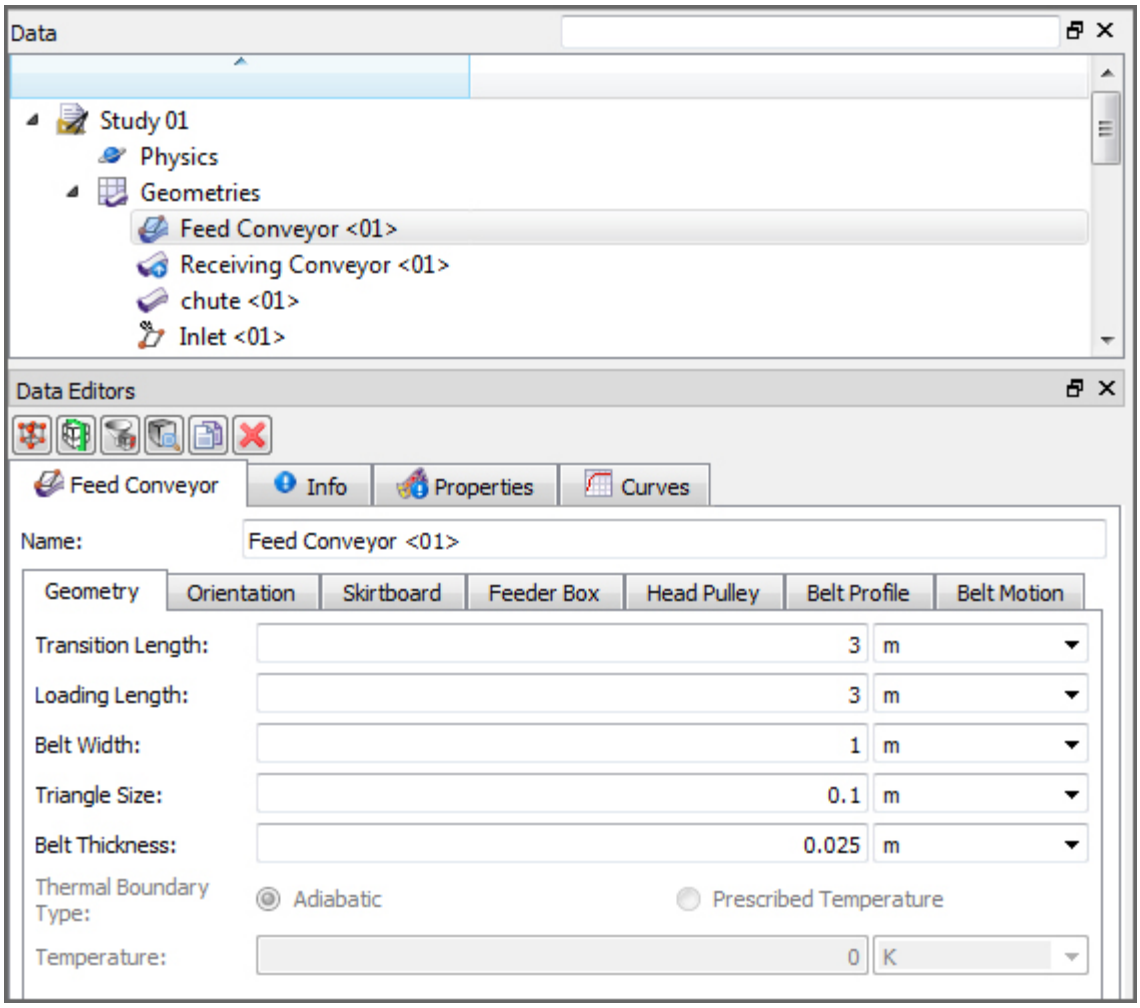
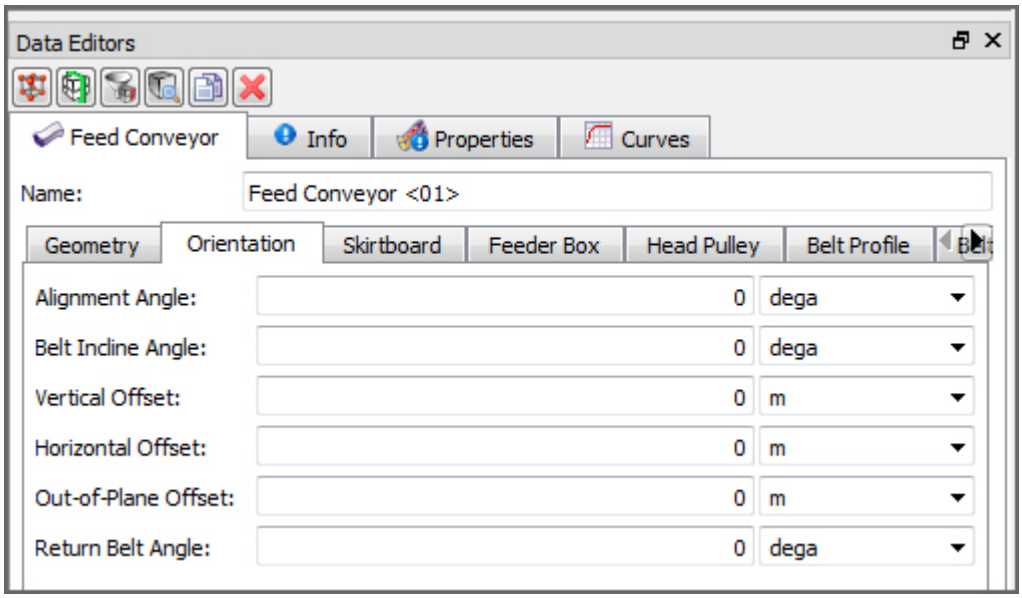


Figure 3.46: Default Feed Conveyor, Orientation parameters in the Data Editors panel



Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

Figure 3.47: Default Feed Conveyor, Skirtboard parameters in the Data Editors panel

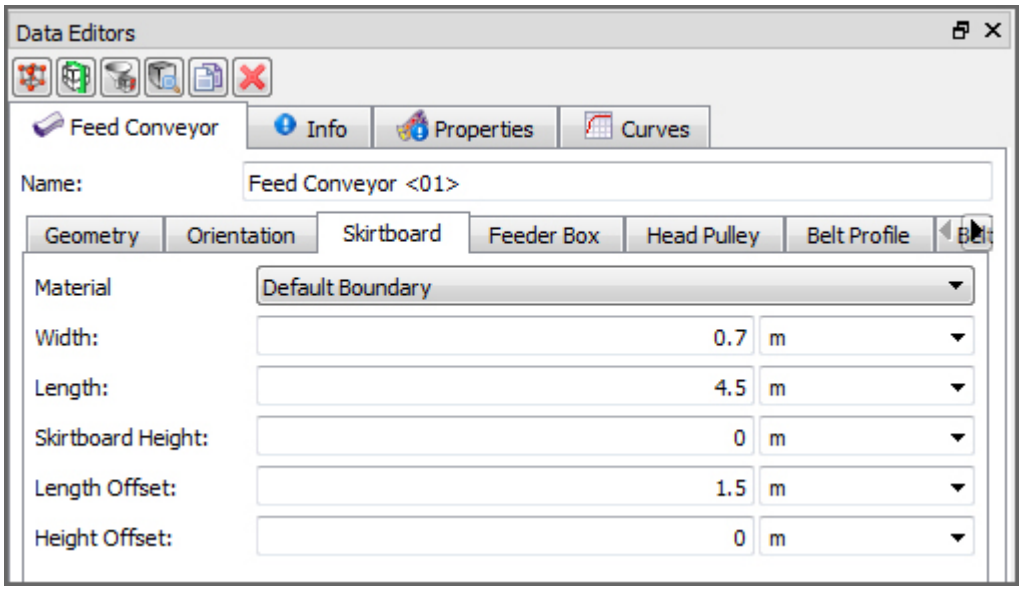


Figure 3.48: Default Feed Conveyor, Feeder Box parameters in the Data Editors panel

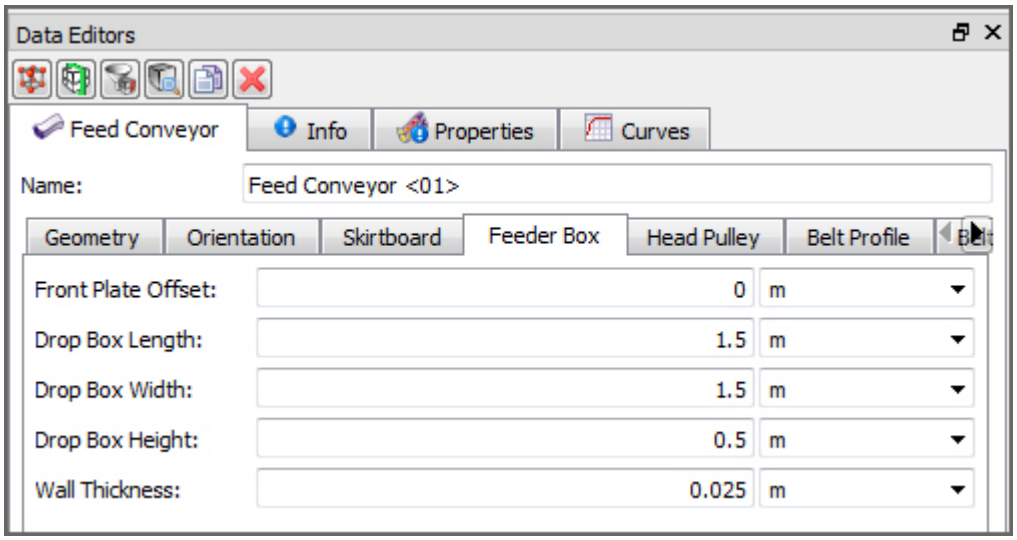
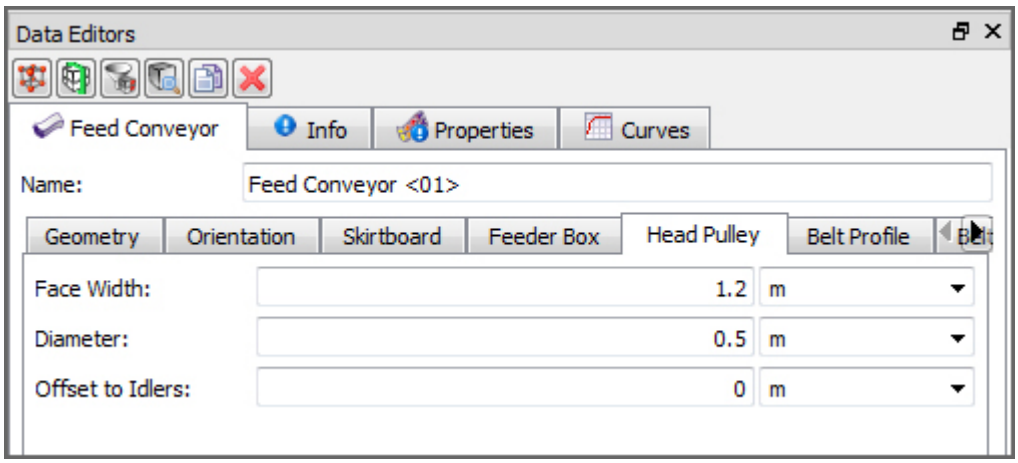


Figure 3.49: Default Feed Conveyor, Head Pulley parameters in the Data Editors panel



Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

Figure 3.50: Default Feed Conveyor, Belt Profile parameters in the Data Editors panel

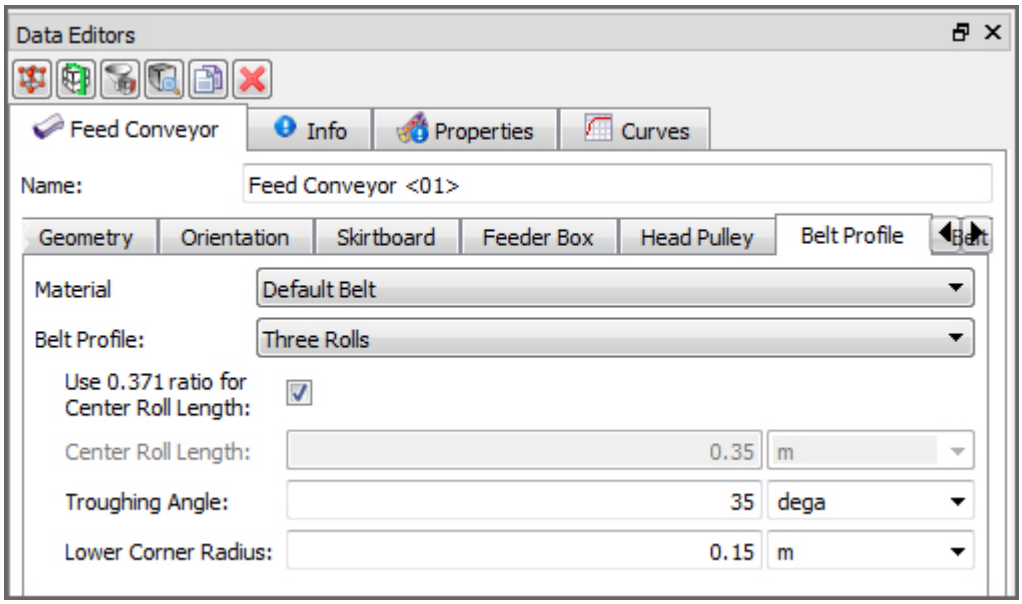


Figure 3.51: Default Feed Conveyor, Belt Motion parameters in the Data Editors panel

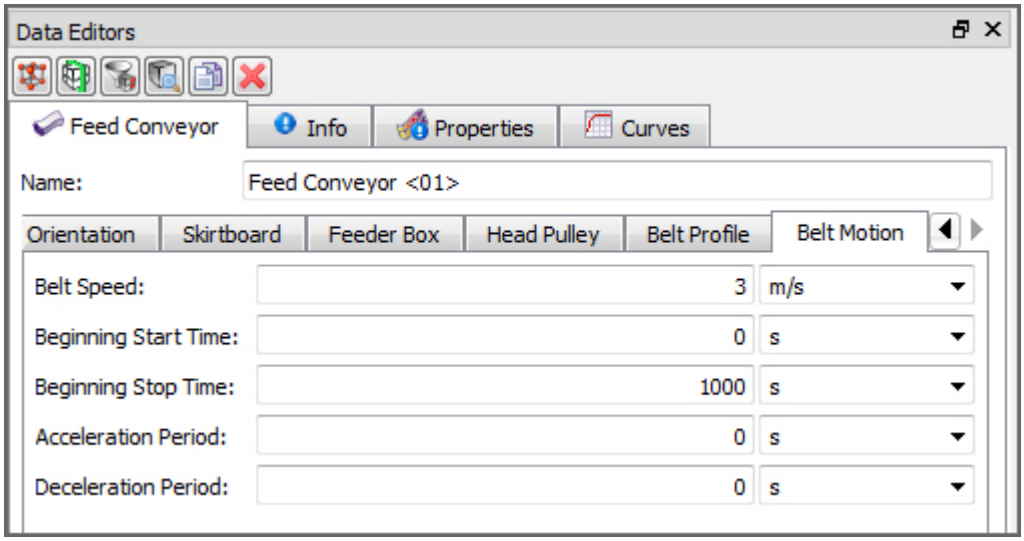


Figure 3.52: Default Feed Conveyor, Modules tab in the Data Editors panel

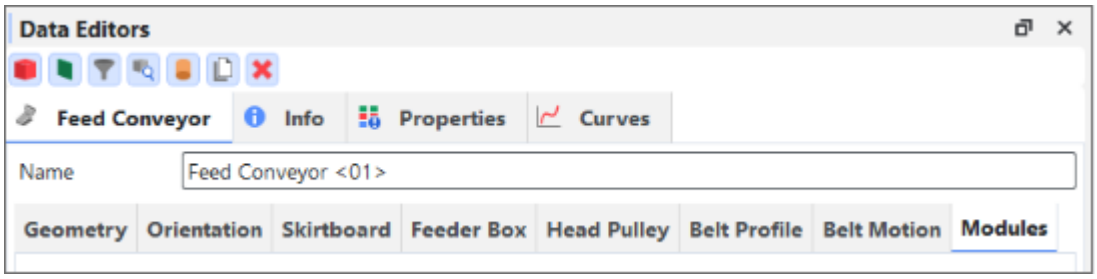


Table 1: Default Feed Conveyor parameter options (all tabs)

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

Setting	Description	Range
Name	Enables you to specify a unique identifier for the geometry component.	No limit
Geometry		
Transition Length	Length of the transition or ending portion of the conveyor belt.	Positive values
Loading Length	Length of the loading or beginning portion of the conveyor belt.	Positive values
Belt Width	Width of the conveyor belt.	Positive values
Triangle Size	Size of the triangular components into which the boundary is divided. This value is used both for refining the boundary mesh (this will not happen if the default Triangle Size is already finer than this value) and for averaging purposes during wear calculations. Therefore, in general, the smaller the Triangle Size, the more accurate the wear calculations but the longer the processing time. (See also View a Color Map of Wear on the Default Belt or Imported Geometry Itself (p. 381).)	Positive values
Belt Thickness	Thickness of the conveyor belt. Affects visualization only; does not affect calculations.	Positive values
Thermal Boundary Type	When **Thermal Model** is enabled (see also About Physics Parameters (p. 417)), this determines how heat conduction is calculated for the boundary. Specifically: • **Adiabatic** applies no heat transfer to the boundary. • **Prescribed Temperature** applies a constant temperature value to the boundary, as specified by the **Temperature** parameter.	Adiabatic; Prescribed Temperature
Temperature	When **Prescribed Temperature** is selected, this sets the temperature value for the boundary.	Values above but not equal to 0 K (zero Kelvin)
Orientation		
Alignment Angle	Angle the top (carry) part of the conveyor will be placed on the horizontal plane.	No limit
Belt Incline Angle	Angle the conveyor will be placed on the vertical plane.	No limit
Vertical Offset	Distance away from zero on the Y axis that the conveyor will be placed.	No limit
Horizontal Offset	Distance away from zero on the X axis that the conveyor will be placed.	No limit

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

Setting	Description	Range
Out-of-Plane Offset	Distance away from zero on the Z axis that the conveyor will be placed.	No limit
Return Belt Angle	Angle the bottom (return) part of the conveyor will be placed on the horizontal plane.	No limit
Skirtboard		
Material	Defines the density and Young's Modulus of the skirtboard based upon the options you have set in the Materials list. (See About Modifying Material Compositions (p. 591) for more information.)	List is based upon the Materials that have been defined
Width	Width of the conveyor skirtboard.	Positive values
Length	Length of the conveyor skirtboard.	Positive values
Skirtboard Height	Height of the conveyor skirtboard.	Positive values
Length Offset	Horizontal distance away from the beginning of the belt to place the skirtboard.	Positive values
Height Offset	Vertical distance away from the belt to place the skirtboard.	Positive values
Feeder Box		
Front Plate Offset	Length that the front plate will extend from the edge of the feeder box over the conveyor belt.	Positive values
Drop Box Length	Length of the feeder box.	Positive values
Drop Box Width	Width of the feeder box.	Positive values
Drop Box Height	Height of the feeder box.	Positive values
Wall Thickness	Thickness of the feeder box and skirtboard walls.	Positive values
Head Pulley		
Face Width	Width of the conveyor pulley.	Positive values
Diameter	Diameter of the conveyor pulley.	Positive values
Offset to Idlers	Vertical distance away from the belt that the idlers will be placed.	No limit
Belt Profile		
Material	Defines the density and Young's Modulus of the belt based upon the options you have set in the Materials list. (See About Modifying Material Compositions (p. 591) for more information.)	List is based upon the Materials that have been defined
Belt Profile	Amount of rolls that support the conveyor belt.	1, 2, 3, or 5 rolls
Use 0.371 ratio for Center Roll Length	When 3 rolls are selected, automatically calculates center roll length as 0.371 of the **Belt Width**.	Turns on or off
Center Roll Length	When 3 or 5 rolls are selected, this specifies the length of the center roll.	Positive values

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

Setting	Description	Range
Troughing Angle	The degree by which the belt trough is formed between the side and center rolls.	Above but not equal to zero (0) degrees. Tip:The recommended range is 0 < 90 degrees.
Lower Corner Radius	Radius of the inside corner between the center and side rolls. The bigger the value, the smoother the transition.	Positive values
Side Roll Length	When 5 rolls are selected, this specifies the length of the two side rolls closest to the center roll.	Positive values
Use 0.223 ratio for Roll Lengths	When 5 rolls are selected, automatically calculates the side roll length as 0.223 of the **Belt Width**.	Turns on or off
Last Roll Angle	When 5 rolls are selected, this specifies the degree by which the belt trough is formed between the side and outer rolls.	0 < 90 degrees
Upper Corner Radius	When 5 rolls are selected, this specifies the radius of the inside corner between the side and outer rolls. The bigger the value, the smoother the transition.	Positive values
Belt Motion		
Belt Speed	Full speed of the conveyor belt after it completes the **Acceleration Period**.	Positive values
Beginning Start Time	The time you want the belt to begin moving. This is when the **Acceleration Period** begins.	Positive values
Beginning Stop Time	The time you want the belt to begin stopping. This is when the **Deceleration Period** begins.	Positive values
Acceleration Period	The length of time you want the belt to take to reach full **Belt Speed**. Acceleration starts at the **Beginning Start Time** and ends after the time period you specify here.	Positive values
Deceleration Period	The length of time you want the belt to take to come to a full stop. Deceleration starts at the **Beginning Stop Time** and ends after the time period you specify here.	Positive values
Modules		
(Varies)	These settings are specific to only certain *external Modules* (p. 1391) and are therefore not documented in the Rocky User Manual. Refer to the Module's documentation (if provided) for more information. (See also Rocky Simulation Entities that can be Affected by Modules (p. 223).)	(Varies)

Adding a Receiving Conveyor Template

Receiving Conveyors can be adjusted to account for different belt and roll configurations, and more.

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

Receiving Conveyor Parameters

Use the following figures and table to understand the various parameters that you can set for a default receiving conveyor.

Figure 3.53: Default Receiving Conveyor, Geometry parameters in the Data Editors panel

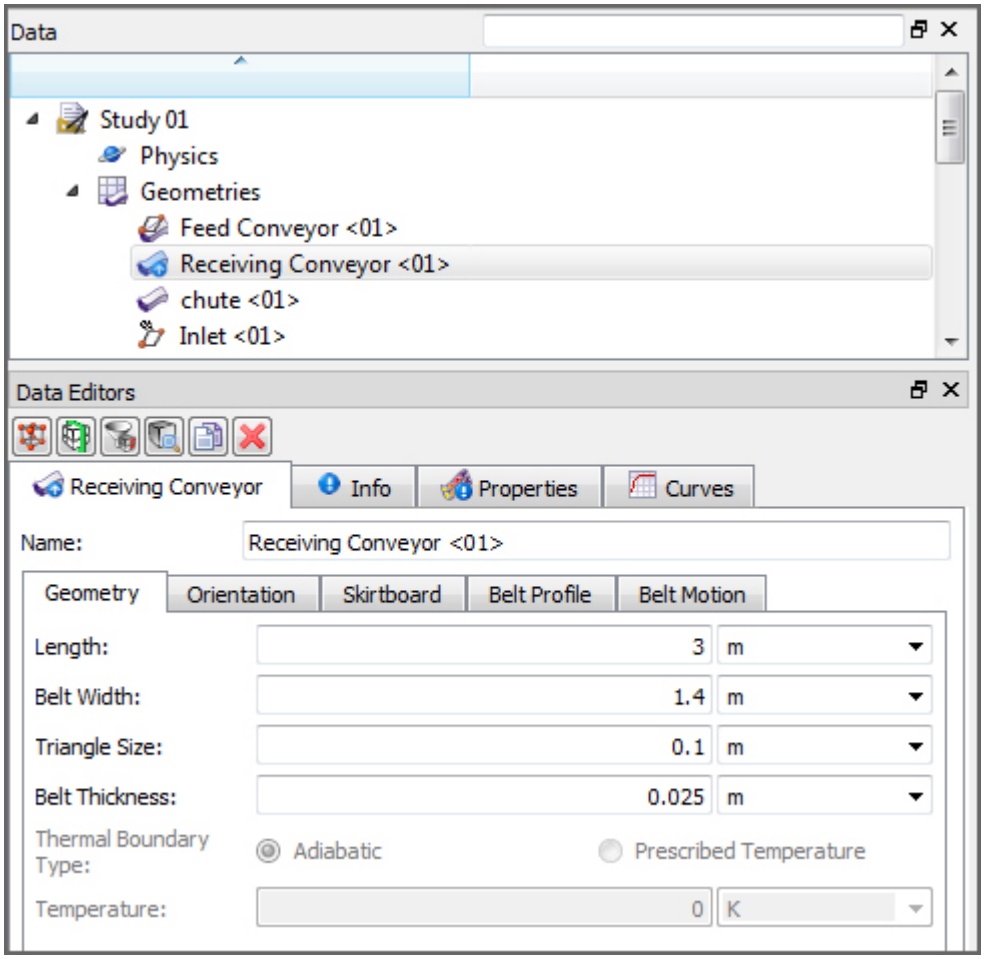
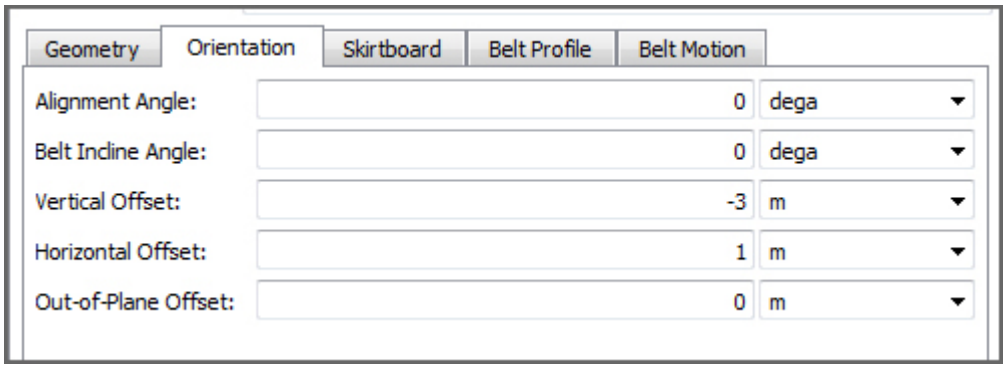


Figure 3.54: Default Receiving Conveyor, Orientation parameters in the Data Editors panel



Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

Figure 3.55: Default Receiving Conveyor, Skirtboard parameters in the Data Editors panel

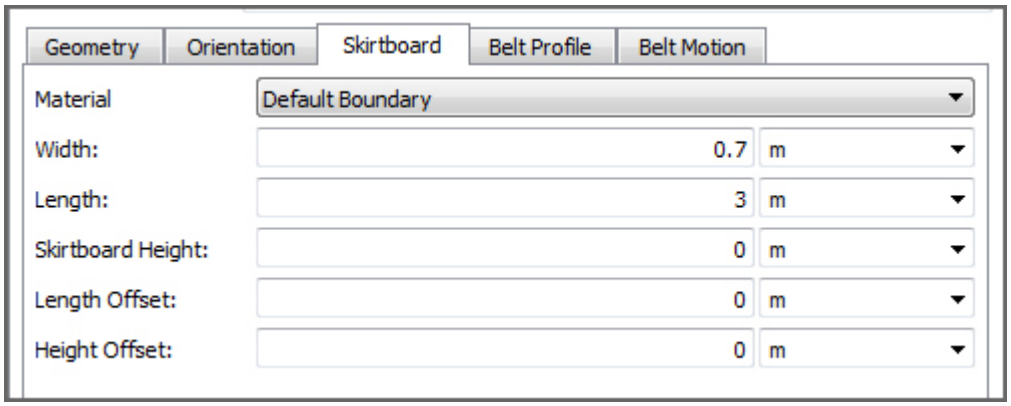


Figure 3.56: Default Receiving Conveyor, Belt Profile parameters in the Data Editors panel

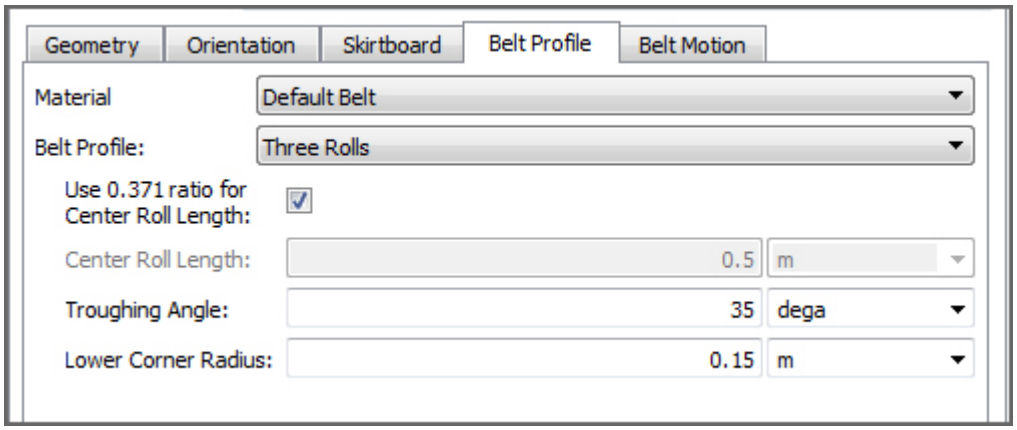


Figure 3.57: Default Receiving Conveyor, Belt Motion parameters in the Data Editors panel

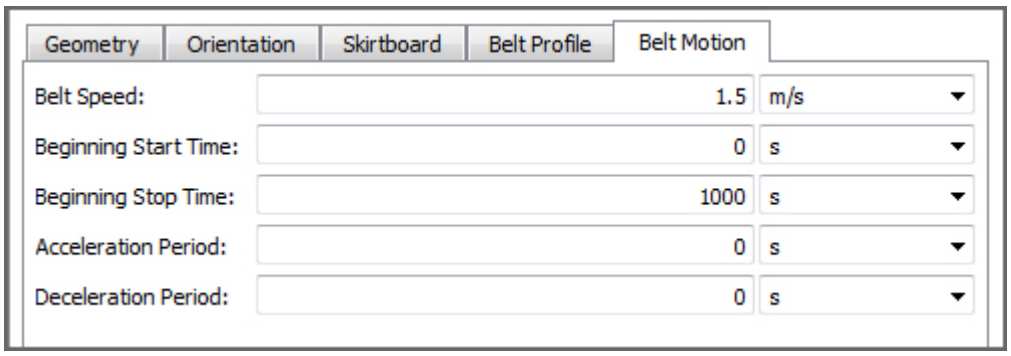


Figure 3.58: Default Receiving Conveyor, Modules tab in the Data Editors panel

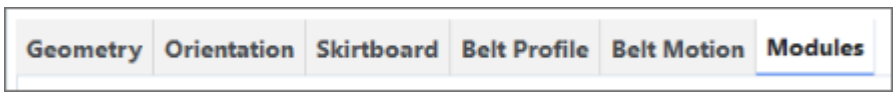


Table 2: Default Receiving Conveyor parameter options (all tabs)

Setting	Description	Range
Name	Enables you to specify a unique identifier for the geometry component.	No limit

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

Setting	Description	Range
Geometry		
Length	Length of the conveyor belt.	Positive values
Belt Width	Width of the conveyor belt.	Positive values
Triangle Size	Size of the triangular components into which the boundary is divided. This value is used both for refining the boundary mesh (this will not happen if the default Triangle Size is already finer than this value) and for averaging purposes during wear calculations. Therefore, in general, the smaller the Triangle Size, the more accurate the wear calculations but the longer the processing time. (See also View a Color Map of Wear on the Default Belt or Imported Geometry Itself (p. 381).)	Positive values
Belt Thickness	Thickness of the conveyor belt. Affects visualization only; does not affect calculations.	Positive values
Thermal Boundary Type	When **Thermal Model** is enabled (see also About Physics Parameters (p. 417)), this determines how heat conduction is calculated for the boundary. Specifically: • **Adiabatic** applies no heat transfer to the boundary. • **Prescribed Temperature** applies a constant temperature value to the boundary, as specified by the **Temperature** parameter.	Adiabatic; Prescribed Temperature
Temperature	When **Prescribed Temperature** is selected, this sets the temperature value for the boundary.	Values above but not equal to 0 K (zero Kelvin)
Orientation		
Alignment Angle	Angle the top (carry) part of the conveyor will be placed on the horizontal plane.	No limit
Belt Incline Angle	Angle the conveyor will be placed on the vertical plane.	No limit
Vertical Offset	Distance away from zero on the Y axis that the conveyor will be placed.	No limit
Horizontal Offset	Distance away from zero on the X axis that the conveyor will be placed.	No limit
Out-of-Plane Offset	Distance away from zero on the Z axis that the conveyor will be placed.	No limit
Skirtboard		
Material	Defines the density and Young's Modulus of the skirtboard based upon the options you've set in	List is based upon the

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

Setting	Description	Range
	the Materials list. (See About Modifying Material Compositions (p. 591) for more information.)	Materials that have been defined
Width	Width of the conveyor skirtboard.	Positive values
Length	Length of the conveyor skirtboard.	Positive values
Skirtboard Height	Height of the conveyor skirtboard.	Positive values
Length Offset	Horizontal distance away from the beginning of the belt to place the skirtboard.	Positive values
Height Offset	Vertical distance away from the belt to place the skirtboard.	Positive values
Belt Profile		
Material	Defines the density and Young's Modulus of the belt based upon the options you've set in the Materials list. (See About Modifying Material Compositions (p. 591) for more information.)	List is based upon the Materials that have been defined
Belt Profile	Amount of rolls that support the conveyor belt.	1, 2, 3, or 5 rolls
Use 0.371 ratio for Center Roll Length	When 3 rolls are selected, automatically calculates center roll length as 0.371 of the **Belt Width**.	Turns on or off
Center Roll Length	When 3 or 5 rolls are selected, this specifies the length of the center roll.	Positive values
Troughing Angle	The degree by which the belt trough is formed between the side and center rolls.	Above but not equal to zero (0) degrees. Tip:The recommended range is 0 < 90 degrees.
Lower Corner Radius	Radius of the inside corner between the center and side rolls. The bigger the value, the smoother the transition.	Positive values
Side Roll Length	When 5 rolls are selected, this specifies the length of the two side rolls closest to the center roll.	Positive values
Use 0.223 ratio for Roll Lengths	When 5 rolls are selected, automatically calculates the side roll length as 0.223 of the **Belt Width**.	Turns on or off
Last Roll Angle	When 5 rolls are selected, this specifies the degree by which the belt trough is formed between the side and outer rolls.	0 < 90 degrees
Upper Corner Radius	When 5 rolls are selected, this specifies the radius of the inside corner between the side and outer rolls. The bigger the value, the smoother the transition.	Positive values
Belt Motion		

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

Setting	Description	Range
Belt Speed	Full speed of the conveyor belt after it completes the **Acceleration Period**.	Positive values
Beginning Start Time	The time you want the belt to begin moving. This is when the **Acceleration Period** begins.	Positive values
Beginning Stop Time	The time you want the belt to begin stopping. This is when the **Deceleration Period** begins.	Positive values
Acceleration Period	The length of time you want the belt to take to reach full **Belt Speed**. Acceleration starts at the **Beginning Start Time** and ends after the time period you specify here.	Positive values
Deceleration Period	The length of time you want the belt to take to come to a full stop. Deceleration starts at the **Beginning Stop Time** and ends after the time period you specify here.	Positive values
Modules		
(Varies)	These settings are specific to only certain *external Modules* (p. 1391) and are therefore not documented in the Rocky User Manual. Refer to the Module's documentation (if provided) for more information. (See also Rocky Simulation Entities that can be Affected by Modules (p. 223).)	(Varies)

What would you like to do?

- Edit the Parameters for a Geometry (p. 519)
- Learn Best Practices for Setting up Geometries (p. 70)

See Also:

- Add Geometry Components (p. 480)
- Set Simulation Parameters (p. 414)
- Add New Circular and Rectangular Surfaces (p. 483)

EDIT THE PARAMETERS FOR A GEOMETRY

1. From the **Data** panel, under **Geometries**, select the name of the geometry that you want to edit. The parameters for that geometry are displayed in the **Data Editors** panel. The tab named with the type of geometry you selected (for example, Wall, "Surface", and so on) will be active.
2. From the **Data Editors** panel, enter the information you want on the active tab, being sure to select each of the various sub-tabs or dialogs that contain parameters you want to modify.

Tip:

To set the same value for a parameter across multiple similar geometries, multi-select the geometries you want in the **Data** panel (SHIFT + left-click for a continuous group; CTRL +

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

left-click for discontinuous items) and then change the values you want in the **Data Editor** panel. Only those parameters common across all selected geometries will be editable but any changes made will populate across the selected group.

See Also:

- Add a Conveyor Template (p. 507)
- Add Geometry Components Parameters (p. 480)
- Add New Circular and Rectangular Surfaces (p. 483)
- Add Geometry Components (p. 480)

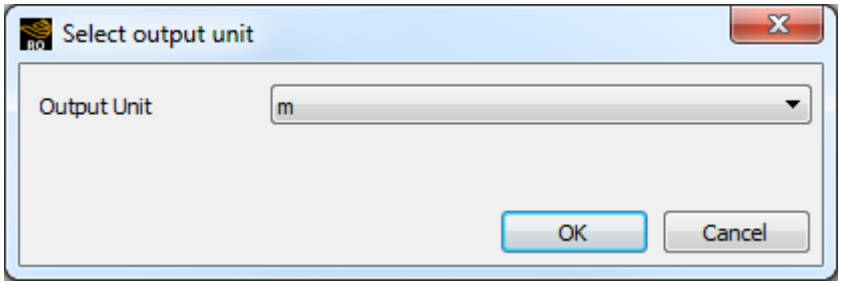
EXPORT A GEOMETRY COMPONENT TO AN STL FILE

In cases where you need to analyze or otherwise make use of your rendered geometry (p. 1394) component outside of Rocky, you may choose to export it to an .stl file. This applies to default Feed Conveyors, default Receiving Conveyors, and any geometry components that you have previously imported. Exporting a geometry at different times for further analysis is especially useful after your geometry surface has been modified by wear. (See also View Surface Wear Modification on an Imported Geometry (p. 379).)

This export ability applies also to any User Processes created from a geometry component, which exports the shape of any (whole) geometry triangles that are selected by the User Process. (See also Export User Process Geometry Triangles to an STL File (p. 1220).)

Exporting a geometry component enables you to select which units you want to use when exporting (Figure 1).

Figure 3.59: Select Output Unit dialog



To export a geometry component to an STL file:

1. If exporting worn geometries, ensure that you have selected what **time step** you want from the **Time** toolbar.
2. From the **Data** panel, under **Geometries**, right-click the component you want to export, point to **Export**, and then click **Rendered Geometry**.
3. From the **Select output unit** dialog, select from the **Output Unit** list the units you want, and then click **OK**.

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

4. From the **Select target STL file** dialog, click the drive or folder of the location to which you want to save the file.
5. In the **File name** box, enter a name for the file, and then click **Save**.

See Also:

- Import Wall or Surface Geometries (p. 487)
- View Surface Wear Modification on an Imported Geometry (p. 379)
- About Exporting Data and Images Out of Rocky (p. 1155)
- Export User Process Geometry Triangles to an STL File (p. 1220)

To export a imported surface to an STL file:

If you import a surface into Rocky, you can also export it following the steps in Geometries Parameters (p. 520). However, it is not possible to export surfaces created inside Rocky.

REMOVE A GEOMETRY

- From the **Data** panel, under **Geometries**, right-click the name of the geometry you want to remove, and then click **Remove Geometry**.

Tip:

If you want to change out a geometry component for a different file, you can also choose to just replace the file. (See also Replace an Imported Geometry File (p. 521).)

See Also:

- Add a Conveyor Template (p. 507)
- Edit the Parameters for a Geometry (p. 519)
- Replace an Imported Geometry File (p. 521)
- Add Geometry Components (p. 480)

REPLACE AN IMPORTED GEOMETRY FILE

1. From the **Data** panel, under **Geometries**, select the imported geometry component you want to replace.
2. From the **Data Editors** panel, select the **Geometry** tab and then click the **Load File** button.
3. From the **Select file to import** dialog, locate and select the geometry file you want to re-import, and then click **Open**.
4. From the **File Import Info** dialog, choose the import options you want, and then click **OK**

See Also:

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

- Add a Conveyor Template (p. 507)
- Add Geometry Components Parameters (p. 480)
- Add New Circular and Rectangular Surfaces (p. 483)
- Add Geometry Components (p. 480)
- Edit the Parameters for a Geometry (p. 519)
- Replace an Imported Geometry File (p. 521)

3.2.9. Create and Edit Regions of Interest

Regions of Interest enable you to specify an area inside the simulation domain within which certain external Modules (p. 1391) can perform custom calculations.

Tip: Unless you have enabled an external Module (p. 1391) that makes use of a Region of Interest, you can skip this part of your Rocky setup.

What would you like to do?

- Learn more About Regions of Interest (p. 522)
- Learn more About the Cube Region of Interest (p. 523)
- Learn more About the Cylinder Region of Interest (p. 526)
- Add a Region of Interest (p. 531)
- Edit or Remove a Region of Interest (p. 532)

See Also:

- About Modules Parameters (p. 431)
- Set Simulation Parameters (p. 414)

ABOUT REGIONS OF INTEREST

A Region of Interest (sometimes referred to as a "ROI") enables you to specify an area inside the simulation domain within which certain external Modules (p. 1391) can perform custom calculations. The calculations within this "sub-domain" results in data that can be visualized and analyzed through other entities (p. 1395)—such as Particles and Geometries—similar to how calculations in the main domain function.

Tip:

Unless you have enabled an external Module that makes use of a Region of Interest, you can skip this part of your Rocky setup.

In this version of Rocky, there are two shape options you can define as a Region of Interest:

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

- A **Cube**, which enables you to create a box-like shape.
- A **Cylinder**, which enables you to create a full or partial cylindrical shape, with or without a center hole.

While both shapes have setups and appearances similar to those of their User Process cousins (see also Analyze Data within a Defined Shape (p. 1175)), a Region of Interest shape is quite different in purpose and function as illustrated in Figure 1.

Figure 3.60: Comparison of a User Process Shape with a Region of Interest Shape

Shape Comparison

	User Process	Region of Interest
Setup Phase	Post-Processing	Pre-Processing
Affects	Data already calculated	Data to be calculated
Applied to	Particles, Geometries, CFD Coupling, other User Processes	External Modules
Abilities	Visualization, Analyitics	Calculation Limits

What would you like to do?

- Learn more About the Cube Region of Interest (p. 523)
- Learn more About the Cylinder Region of Interest (p. 526)
- Add a Region of Interest (p. 531)
- Edit or Remove a Region of Interest (p. 532)

See Also:

- About Modules Parameters (p. 431)
- About Using the Coloring Tab to Change a 3D View (p. 1074)
- View Geometries, Particles, Points, and Fluids in 3D (p. 1041)

ABOUT THE CUBE REGION OF INTEREST

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

The Cube Region of Interest enables you to define within the simulation boundaries a three dimensional box of whatever length, width, and height you want. In this way, you can limit the calculations of certain external Modules (p. 1391) to only the domain selection contained within the box.

Tip:

Unless you have enabled an external Module that makes use of a Region of Interest, you can skip this part of your Rocky setup.

Like Cube User Processes, you can use the colored directional handles (Figure 1) in a 3D View window to change the shape and location of the box (see also Use Colored Directional Handles (p. 126)), or you can define exact values in the Data Editors panel.

See the image and table below to learn more about the Cube Region of Interest.

Cube Region of Interest Parameter Definitions

Figure 3.61: Cube Region of Interest during project setup

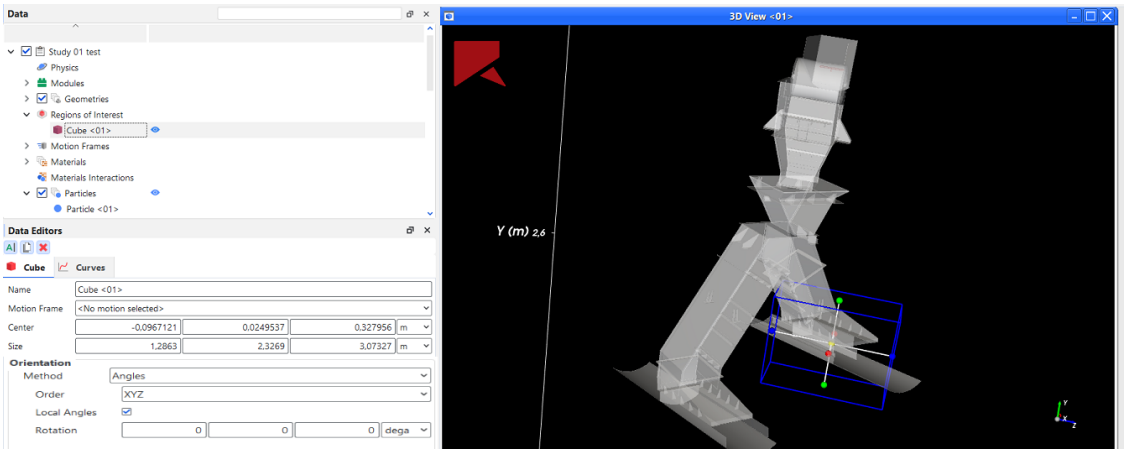


Table 1: Cube Region of Interest parameter options

Setting	Description	Range
Name	Enables you to specify a unique identifier for the Region of Interest. **Note:** This same name will appear in the **Data** panel under **Regions of Interest**.	No limit
Center	The X, Y, and Z coordinate locations of the cube's center point.	Any value
Size	The length of the cube's three sides as specified in the X, Y, and Z directions.	Any value
Orientation		

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

Setting	Description	Range
Method	Enables you to select how you want to define the orientation of the cube shape. Specifically: • **Angles** enables you to define angles of rotation in three directions, the order of which you can also specify. • **Angle and Vector** enables you define one vector and one angle of rotation around it. • **Basis Vectors** enables you to define the X, Y, and Z directions of the cube's local basis.	Angles; Angle and Vector; Basis Vectors
Method Angles		
Order	When **Angles** is selected for **Orientation**, this defines the order in which the three **Rotation** text fields will be applied.	XYZ; ZXY; YXZ; YZX; ZXY; ZYX
Local Angles	When **Angles** is selected for **Orientation**, this defines what coordinate system will be used as a basis for the angle specified. Specifically: • When enabled, the angle will be based on the local coordinate system. • When cleared, the angle will be based on the global coordinate system.	Turns on or off
Rotation	When **Angles** is selected for **Orientation**, this is the degree of cube rotation in each of the three directions specified by the **Order** provided.	Any value
Method Angle and Vector		
Angle	When **Angle and Vector** is selected for **Orientation**, this is the angle the cube will rotate around the **Vector** defined.	Any value
Vector	When **Angle and Vector** is selected for **Orientation**, this is the X, Y, and Z components that define the vector around which the cube will rotate, using the **Angle** defined.	No limit but the values entered will be normalized
Method Basis Vectors		
X direction	When **Basis Vectors** is selected for **Orientation**, this is the coordinate values defining the first of three	No limit but the values entered will be normalized

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

Setting	Description	Range
	directional vectors that together define the final orientation of the cube. **Tip:** To ensure correct results, make sure you define this vector as orthogonal (perpendicular) to the other two vectors.	
Y direction	When **Basis Vectors** is selected for **Orientation**, this is the coordinate values defining the second of three directional vectors that together define the final orientation of the cube. **Tip:** To ensure correct results, make sure you define this vector as orthogonal (perpendicular) to the other two vectors.	No limit but the values entered will be normalized
Z direction	When **Basis Vectors** is selected for **Orientation**, this is the coordinate values defining the third of three directional vectors that together define the final orientation of the cube. **Tip:** To ensure correct results, make sure you define this vector as orthogonal (perpendicular) to the other two vectors.	No limit but the values entered will be normalized

What would you like to do?

- Add a Region of Interest (p. 531)
- Edit or Remove a Region of Interest (p. 532)

See Also:

- About Regions of Interest (p. 522)
- Use Colored Directional Handles (p. 126)

ABOUT THE CYLINDER REGION OF INTEREST

The Cylinder Region of Interest enables you to define within the simulation boundaries a three dimensional analysis tube, arch, or slice of whatever radius and height you want, and with or without a center hole. (See Figure 3 below for examples.) In this way, you can limit the data an external Mod-

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

ules (p. 1391) calculates to only the selection contained by the shape you determine. The method is particularly useful for analyzing certain areas of equipment with round components, such as mills and pipes.

Note:

Unlike Cylinder User Processes, which can be elliptical in shape (see also About the Cylinder User Process (p. 1181)), Region of Interest Cylinders must only be perfectly round.

See the images and table below to learn more about the Cylinder Region of Interest.

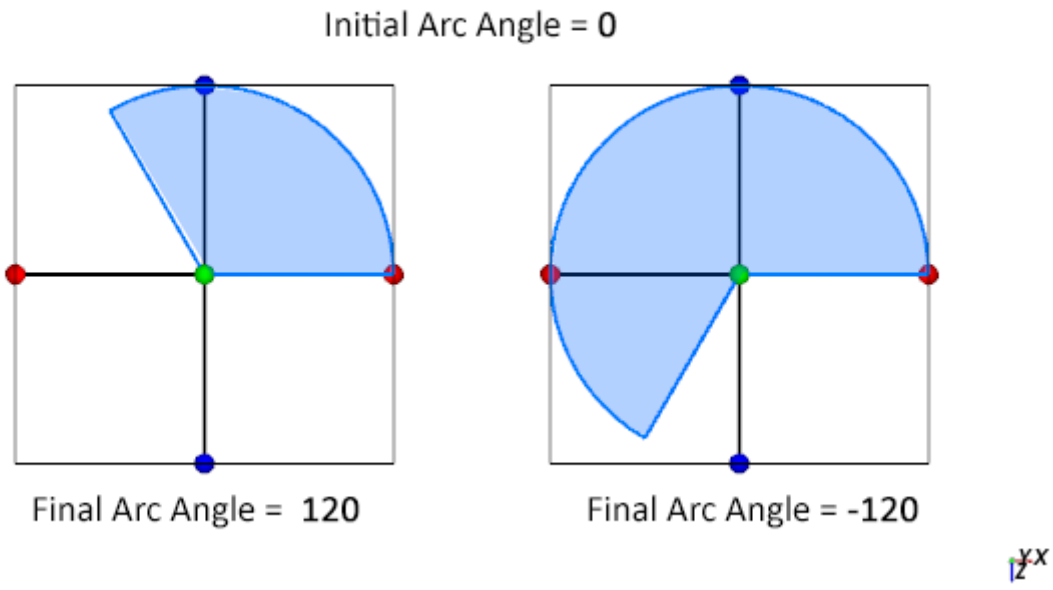
Tip:

You can Use Colored Directional Handles (p. 126) to change the shape and location.

About Cylinder Arc Angles

What you set for your **Initial** and **Final** Arc Angle values determine how far around the Y axis the arc for the cylinder will be drawn. Cylinder Arc Angles follow the right-hand rule: when the Y axis points outside the screen (from the **Fit** menu, select **Camera Preset: +Y**), no matter whether you enter a positive or negative value for the **Final** parameter, the arc will be drawn from the **Initial** value counterclockwise to your **Final** value. (See Figure 1 below for an illustration.)

Figure 3.62: Cylinder arc results with different Final angle values



CYLINDER REGION OF INTEREST PARAMETER DEFINITIONS

Figure 3.63: Cylinder Region of Interest

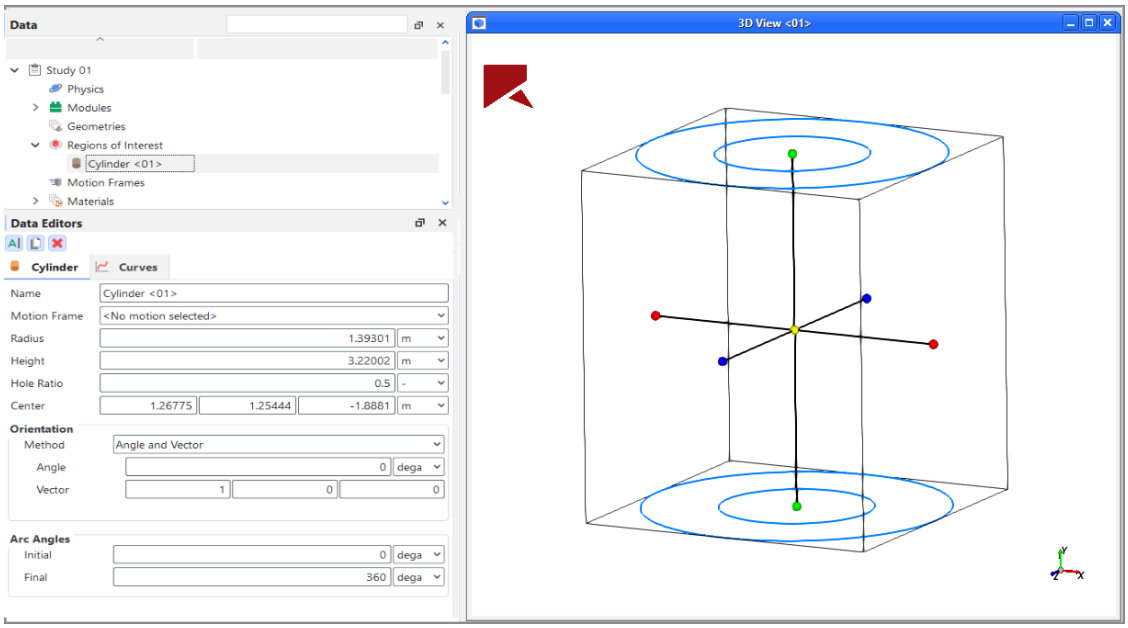


Figure 3.64: Cylinder shape examples: arch (left), slice (center), and tube (right)

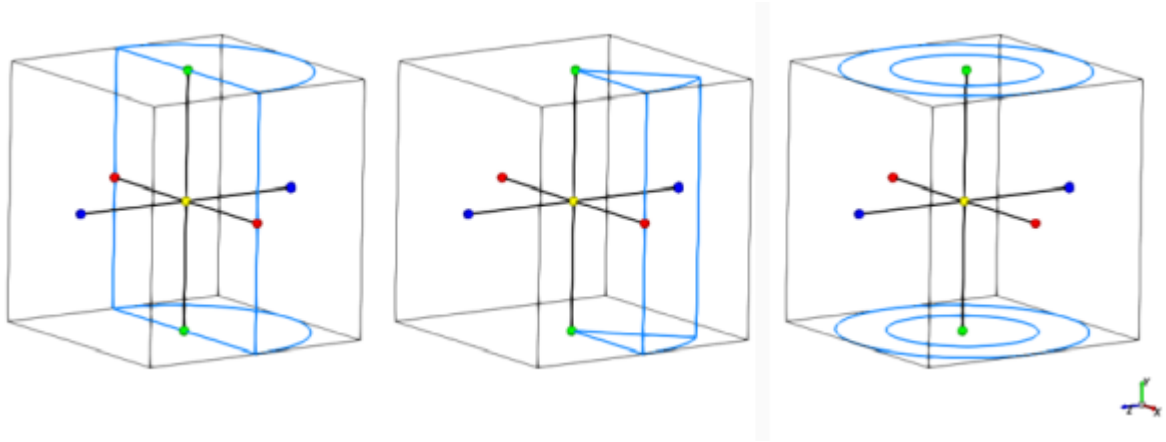


Table 1: Cylinder Region of Interest parameter options

Setting	Description	Range
Name	Enables you to specify a unique identifier for the Region of Interest. **Note:** This same name will appear in the **Data** panel under **Regions of Interest**.	No limit
Center	The X, Y, and Z coordinate locations of the shape's center point.	Any value
Radius	The size of the cylinder's outer radius.	Any value

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

Setting	Description	Range
Height	The size of the shape's dimension in the Y direction.	Any value
Hole Ratio	The percentage of volume that the center hole through the cylinder will take up.	0-1
Orientation		
Method	Enables you to select how you want to define the orientation of the cylinder shape. Specifically: • **Angles** enables you to define angles of rotation in three directions, the order of which you can also specify. • **Angle and Vector** enables you define one vector and one angle of rotation around it. • **Basis Vectors** enables you to define the X, Y, and Z directions of the cylinder's local basis.	Angles; Angle and Vector; Basis Vectors
Method Angles		
Order	When **Angles** is selected for **Orientation**, this defines the order in which the three **Rotation** text fields will be applied.	XYZ; ZXY; YXZ; YZX; ZXY; ZYX
Local Angles	When **Angles** is selected for **Orientation**, this defines what coordinate system will be used as a basis for the angle specified. Specifically: • When enabled, the angle will be based on the local coordinate system. • When cleared, the angle will be based on the global coordinate system.	Turns on or off
Rotation	When **Angles** is selected for **Orientation**, this is the degree of cylinder rotation in each of the three directions specified by the **Order** provided.	Any value
Method Angle and Vector		
Angle	When **Angle and Vector** is selected for **Orientation**, this is the angle the cylinder will rotate around the **Vector** defined.	Any value
Vector	When **Angle and Vector** is selected for **Orientation**, this is the X, Y, and Z components that define the vector around which the cylinder will rotate, using the **Angle** defined.	No limit but the values entered will be normalized
Method Basis Vectors		

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

Setting	Description	Range
X direction	When **Basis Vectors** is selected for **Orientation**, this is the coordinate values defining the first of three directional vectors that together define the final orientation of the cylinder. **Tip:** To ensure correct results, make sure you define this vector as orthogonal (perpendicular) to the other two vectors.	No limit but the values entered will be normalized
Y direction	When **Basis Vectors** is selected for **Orientation**, this is the coordinate values defining the second of three directional vectors that together define the final orientation of the cylinder. **Tip:** To ensure correct results, make sure you define this vector as orthogonal (perpendicular) to the other two vectors.	No limit but the values entered will be normalized
Z direction	When **Basis Vectors** is selected for **Orientation**, this is the coordinate values defining the third of three directional vectors that together define the final orientation of the cylinder. **Tip:** To ensure correct results, make sure you define this vector as orthogonal (perpendicular) to the other two vectors.	No limit but the values entered will be normalized
Arc Angles		
Initial Arc	The initial degree point around the Y axis that the circular portion of the cylinder will be drawn. When the Y axis points outside the screen, the resulting arc will be drawn from the initial angle counterclockwise to the final angle. **Tip:** To draw a full cylinder, keep this value zero (0) and set **Final** to 360.	Any value
Final Arc	The final degree point around the Y axis that the circular portion of the cylinder will be drawn. when the Y axis	Any value

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

Setting	Description	Range
	points outside the screen, the resulting arc will be drawn from the initial angle counterclockwise to the final angle. **Tip:** To draw a full cylinder, set this value to 360 and keep **Initial** at zero (0).	

What would you like to do?

- Add a Region of Interest (p. 531)
- Edit or Remove a Region of Interest (p. 532)

See Also:

- About Regions of Interest (p. 522)
- Use Colored Directional Handles (p. 126)

ADD A REGION OF INTEREST

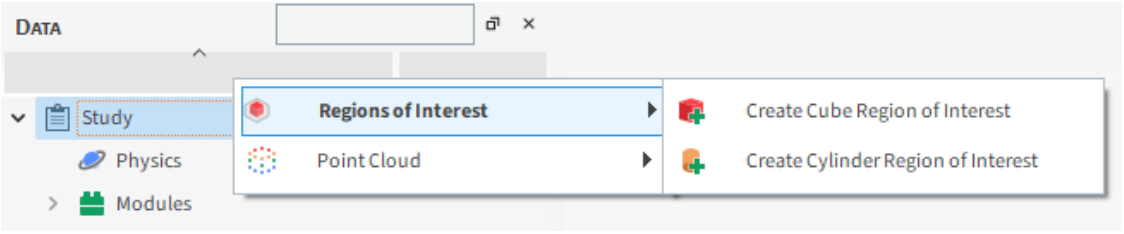
1. Ensure you have enabled an external Module (p. 1391) that makes use of Regions of Interest.

   Tip:

   If you do not plan to enable an external Module that makes use of Regions of Interests, then you do not need to add a Region of Interest to your simulation.

2. From the **Data** panel, right-click **Study**, select **Regions of Interest** and then create the shape you want to.

   Figure 3.65: How to create a Region of Interest



   Note:

   The **Regions of Interest** entity shows up when you create a ROI, otherwise it will be hidden from the Study.

3. From the **Data** panel, select the newly created ROI below **Regions of Interest**, and from the **Data Editors** panel, on the shape tab for the item (**Cube** or **Cylinder**), enter the options you want.

Tip:

In the 3D View window (see also Create and Modify a 3D View (p. 1040)), you can also move and resize the Region of Interest area by clicking and dragging the handles (colored dots). (See also Use Colored Directional Handles (p. 126).)

See Also:

- About Regions of Interest (p. 522)
- About the Cube Region of Interest (p. 523)
- About the Cylinder Region of Interest (p. 526)
- About Modules Parameters (p. 431)

EDIT OR REMOVE A REGION OF INTEREST

1. From the **Data** panel, under **Regions of Interest**, select the entry that you want to change.
2. From the **Data Editors** panel, do one of the following:
   - To edit the entity, from the shape tab for the item (**Cube** or **Cylinder**), change the settings you want.
   - To remove the entity, click the **Remove** button.

See Also:

- About Regions of Interest (p. 522)
- About Modules Parameters (p. 431)
- Add a Region of Interest (p. 531)

3.2.10. Create and Apply Motion Frames

Motion Frames are the method by which you enable geometry components (wall or surfaces)-or when coupled with Particle Custom Inlets to translate, rotate, vibrate, swing, and/or move during your simulation. If a surface is attached to an Inlet/Outlet, this component will also inherit the motion assigned to the geometry. You first define your desired movement within the Motion Frame, and then assign the Motion Frame to the geometry or Particle Custom Inlet you want to move. After setting up and applying your Motion Frames, it is a good idea to preview your motions in a Motion Preview window before processing the simulation.

Because the crushing movement can be more complicated to set up than other kinds of movements, Rocky also enables you to create a Cone Crusher Frame, which works similarly to a Motion Frame but has its own unique parameters.

What would you like to do?

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

- Learn more About Creating and Applying Motion Frames (p. 533)
- Create and Apply Cone Crusher Frames (p. 564)
- Create a Motion Frame (p. 569)
- Apply a Motion Frame to an Imported Geometry (p. 569)
- Apply a Motion Frame to a Particle Custom Inlet (p. 571)
- Create and Modify a Motion Preview Window (p. 571)

See Also:

- About Importing Geometry Components (p. 487)

About Creating and Applying Motion Frames

Creating and applying one or more Motion Frames to an imported geometry-or when coupled with Custom Inputs, the "frozen" portion of a frozen Fiber (p. 1391)-enables that component to move or animate during your simulation. You can create simple movements-including translation, rotation, vibration, and swinging motions-or more complex movements by nesting several Motion Frames, and/or by enabling six degrees of freedom (6DOF) functionality (also known as Free Body Motion). Crushing movements are defined with a separate kind of frame, called a Cone Crusher Frame, but are created, applied, and previewed in the same manner as a standard Motion Frame. (See also About Creating and Applying Cone Crusher Frames to Imported Geometries (p. 565).)

Motion Frames can be set only for imported geometries and Custom Inputs when set with frozen Fiber particle set, but it is important to note that all default conveyors included within Rocky have **Keep in place** (Global) translation values already enabled. This is how the particles appear to move along the conveyor belt even though the conveyor surface itself appears static.

Motion Frames work best in conjunction with the Motion Preview window as it allows you to view and test your motions before processing your simulation. (See also Preview a Motion in 3D (p. 573).

Note:

Motion Frames curves are statistical, which means the results are evaluated at each output, and any information between outputs is not taken into account.

Note:

When a geometry is disabled in Rocky by increasing its enable time (See also Imported Wall Parameters (p. 494).), it is removed from the particle contact search process. However, if the geometry is associated with a motion frame, its position and orientation are still updated in accordance with the motion frame's dynamics. The geometry's

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

mass properties are accounted for in the calculation of motion frame reactions, ensuring consistency in the simulation's overall dynamics.

Important:

When setting up Motion Frames and Geometries, it is important to always remember to check and, if necessary, adjust the Mass properties of the Geometry that receives the Motion Frame. This is necessary so the solver adopts the correct reference frames, regarding the Geometry's Center of Mass, and the Motion Frame's center.

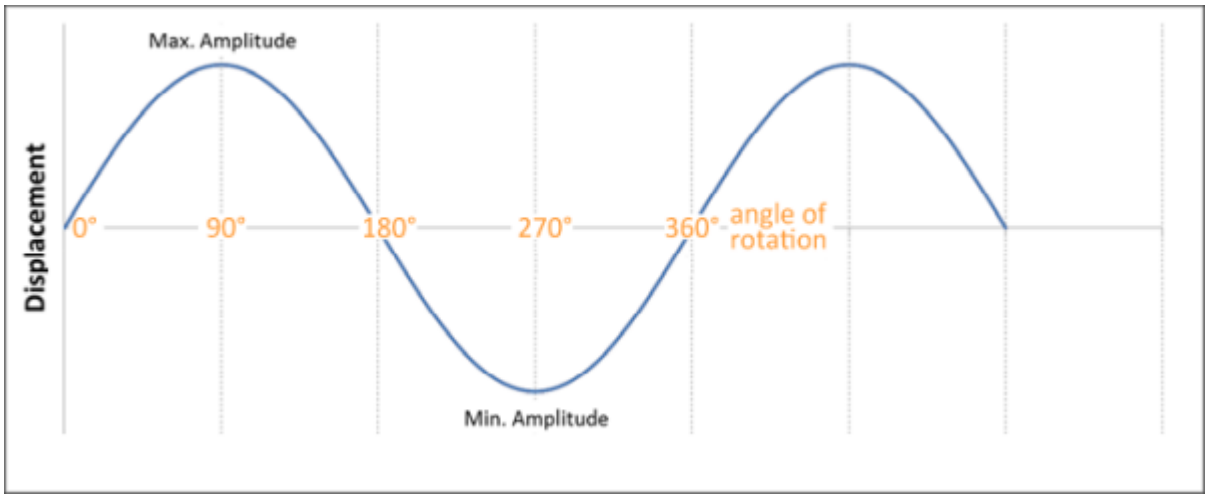
Tip:

If you wish to have more information regarding Motion Frames, please refer to the **Rocky Technical Manual**. There you will find a specific section covering Motion Frames from a technical perspective.

ABOUT INITIAL PHASES (VIBRATION AND PENDULUM MOTIONS)

Periodic Translation (Vibration) and Periodic Rotation (Pendulum) motions are defined by specifying the amplitude and frequency values along a sine wave. The amplitude defines how far from the center point the movement translates (or rotates) and the frequency defines how many complete wave cycles will occur per second. The point along the sine wave period at which the motion begins along the sine wave is defined by the Initial Phase value. An Initial Phase value of zero degrees (default) causes the sine wave to start at the center point of the motion (Figure 1). For a simple linear vibration moving back and forth along the X axis, the frame would start at the zero position, move to the right most limit (maximum amplitude), reverse direction past the zero position to the left most limit (minimum amplitude), and then complete the cycle by reversing direction again to the zero (or 360 degree) position.

Figure 3.66: Sine wave for a vibration when the Initial Phase is zero (default value)

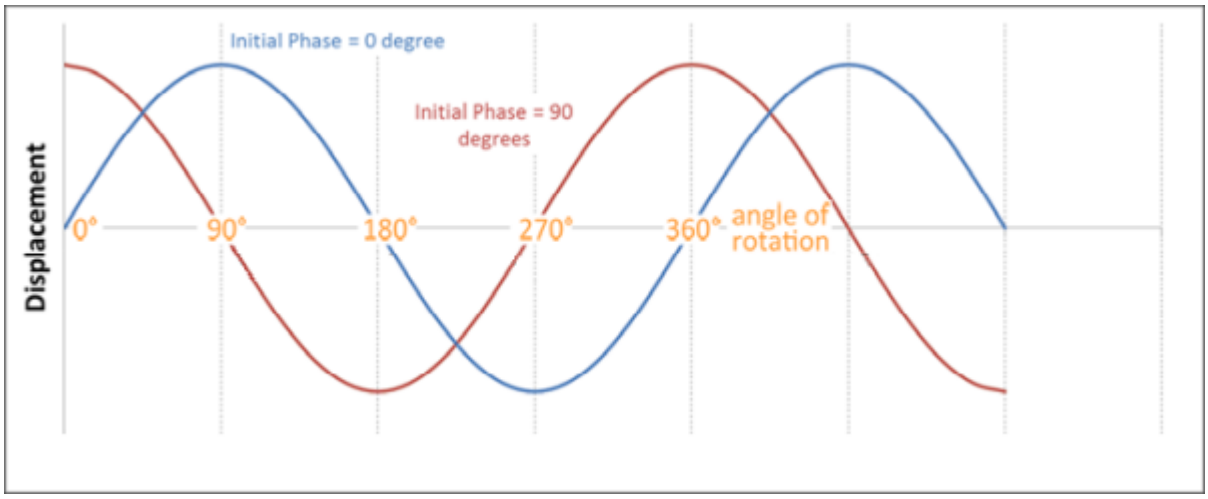


By comparison, changing the Initial Phase value to 90 degrees moves the starting point of the sine wave to the maximum amplitude, or in the simple linear vibration example, the right-most limit along the X axis (Figure 2). In this example, the frame would start its motion at the right-most limit along

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

the X axis, then reverse direction past the zero position to the left most limit (minimum amplitude), and then reverse again to end its cycle at the right-most limit position.

Figure 3.67: Sine wave for a vibration when the Initial Phase is 90 degrees



Changing the Initial Phase can be useful in cases where you want the center of the motion to remain the same but want the motion itself to begin from a non-center position. This is especially useful when combining two motions to create a complex motion.

For example, to create a circular vibration motion along the XY axes, you would create two separate vibration motions: the first along the X axis with Initial Phase at zero degrees, and the second along the Y axis with the Initial Phase at 90 degrees. This enables the vibration along the Y axis to start at its maximum (highest) position at the same time the vibration along the X axis begins its movement to the right from its center point. The combination of these two motions creates the desired circular vibration motion.

Tip:

To see a walk-through example that uses vibration motions, refer to the following Workshop:

Tutorial - Vibrating Screen in the *Rocky Tutorial Guide*.

ABOUT PERIODIC MOTIONS

Periodic motions enables you to loop or repeat a set of motions within your simulation. When Enable Periodic Motion is turned on for a frame, the full list of motions contained within that frame will be repeated as soon as the motion with the last Stop Time completes. The amount of time between the earliest motion's Start Time and the latest motion's Stop Time is saved within Rocky as the periodic motion period.

This periodic motion period is useful to know when setting geometry replication as it can help you define your Replication Interval. To include an evenly spaced copy of the geometry along your peri-

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

odic motion path, the Replication Interval should be equal to the periodic motion period divided by the Number of Replications you have defined.

Tip:

To see a walk-through example that includes both periodic motions and geometry replication, refer to the following Workshop: Tutorial - **Bucket Conveyor** in the *Rocky Tutorial Guide*.

ABOUT FREE BODY MOTIONS

Free body motions, including **Free Body Translation** and **Free Body Rotations**, enable the frame to move freely in response to outside forces. These forces can come from particle contacts, gravity, or from an additional force that you prescribe through the **Additional Force** or **Additional Moment** motions, the **Spring-Dashpot Force** or **Spring-Dashpot Moment** motions, or the **Linear Time Variable Force** or **Linear Time Variable Moment** motions.

Free body motions do not currently respond to other boundary interactions, nor are they directly affected by CFD fluid flow. So setting an object to drop freely onto another object from some height will cause the object to fall through the other object. Having that same object drop freely onto a bed of particles, however, will cause the object to displace the particles and slow the object's falling as expected.

So too with CFD coupling simulations. Setting (on the Rocky side only) a flap-like object affixed at its top to rotate freely against a flow of fluid from the CFD side will cause the fluid to run straight through the flap without any movement. Having that same fluid carry a flow of particles, however, will cause the flowing particles to move the flap due to CFD fluid forces acting upon the particles, and not from the fluid forces over the geometry. Note: Geometries that are represented on both the DEM and CFD sides of a coupling simulation should not use free body motions. Rather, these geometries should use only the **Fluent Motion Frame** that is automatically created when the Fluent case is imported into Rocky. (See also **About Importing Movements from Fluent** section below.)

An **Additional Force**, **Spring-Dashpot Force**, or **Linear Time Variable Force** can be added to a **Free Body Translation** motion in cases where you want an additional force in a given direction to affect the translation, and you want that force to be considered in the motion calculation along with the weight, gravity, particle interactions.

For example, to simulate a car tire rolling along a bed of particles, aside from setting **Free Body Translation** and **Free Body Rotation** motions to simulate the horizontal translation and rotation of the tire, you could also add an **Additional Force** to account for the forward motion of the car itself. In this way, the acceleration and velocity of the translation will be affected by the particles and gravity values: given the same force applied, the car moves slower in sticky mud and faster when on dry road. If only a **Translation** were given instead of a combined **Additional Force** and **Free Body Translation**, the velocity of the translation would remain constant no matter the conditions of the other particles and forces coming into contact with it.

So too, an **Additional Moment**, **Spring-Dashpot Moment**, or **Linear Time Variable Moment** can be added to a **Free Body Rotation** motion in cases where you want an additional moment (or torque) in a given direction to affect only the rotation, and you want that additional torque to be included along with gravity and particle interactions in the motion calculation.

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

Free body motions can also have range limits applied to restrict free movement in a given direction.

Important:

In order to preview a free body motion in the **Motion Preview** window or start processing the simulation, ensure there is a geometry component in the project that has the free body motion associated to it; otherwise, you will get an error.

In this release of Rocky, you are able to add many more types of concurrent free body motions and Frames, including nested Frames, as long as free body motions of the same type (Translation or Rotation) and direction do not overlap in time. Tip:See Geometry Limitations (p. 58) for an updated list of what to be aware of when working with Motion Frames that use Free Body motions.

In this version of Rocky, the Motion Preview window will preview the effects of gravity and any additional (prescribed) forces/moments that you define for free body motions, but will not be able to predict motions as a result of interactions with particles.

Note:

Free body motions are not compatible with geometry replication features.

Tip:

To see walk-through examples that use free body motions, refer to the following Workshops:

Important:

Once the geometry has a free body motion you must correctly define the **Boundary Mass**, **Gravity Center** an **Moment of Inertia** properties to avoid issues.

- Free Body Translation: Tutorial - Drop Weight Test in the *Rocky Tutorial Guide*
- Free Body Rotation and Spring Dashpot Moment: Workshop 06 - High Pressure Grinding Rolls (HPGR)

About Importing Movements from Fluent

2-Way coupling simulations between Ansys Fluent and Rocky (See also About Using the 2-Way Fluent Method (p. 795)) will now support Fluent's Moving Meshes for simple translation and/or simple rotation movements. For geometry movements that are represented on both the DEM and CFD sides, this ability helps ensure that the translation or rotation motions are kept the same in both programs. Multiple zones can be imported.

To enable this ability, the desired translation and/or rotation motion should be represented only on the Fluent side using Moving Meshes. Once these settings are saved into the Fluent CAS (CAS.H5 or CAS.GZ) file, that file is then imported into the Rocky project as part of the Fluent Two Way Coupling

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

process, which then automatically turns the Fluent Moving Meshes information into new Rocky Motion Frame entries, one per Cell Zone with motions you have defined.

Important:

If you decide to use expressions when defining your moving meshes in Fluent, Rocky will support this but will be unable to automatically replicate the Fluent motions for you in Rocky. In these particular cases, you must ensure that you manually create Motion Frames in Rocky that replicate the geometry movement in Fluent as closely as possible.

After the Motion Frames are created (either automatically by Rocky or manually by you), they need to be assigned to the appropriate geometry components within Rocky. These can be identical to the Fluent geometries if they were imported into Rocky from the Fluent CAS, CAS.H5, or CAS.GZ file, or can be completely different geometries that need to maintain the same motions as in Fluent, depending upon your simulation goals.

Unlike other Motion Frames created in Rocky, the parameters of any Fluent Motion Frames that were automatically created by Rocky will be disabled but still visible in the UI. This is to ensure the movements between the two programs are kept the same.

About Nested Motion Frames

In order to create complex motions, Rocky allows you to nest motion frames under other motion frames. By doing so, the nested frame (child) becomes linked to the frame above it in the tree (parent). The child motion frame will move together with the parent motion frame and will also prescribe its own motions.

Because of this, only the child frame in a nested frame situation will be assigned to a geometry, as the child inherits its parent's motions in combination with its own motions.

Figure 3 below shows an example of a two child frames (**Frame <01> <01>** and **Frame <01> <02>**) nested beneath a parent frame (**Frame <01>**).

Tip:

To see a walk-through example of nested motion frames, refer to the following Workshop:

Tutorial - Conical Dryer in the *Rocky Tutorial Guide*.

About Duplicating Motion Frames

Like any other Data panel item, you can duplicate an individual Motion Frame to create an exact copy of it (see also Duplicate a Data Panel Item (p. 846)). This exact copy includes any nested or child Frames that exist underneath the Frame that you are copying.

In Figure 3 below, **Frame <01> <01>** and **Frame <01> <02>** are nested or child Frames of **Frame <01>**. When **Frame <01>** is duplicated, both child Frames are duplicated as well (Figure 4), so that the new **Frame <02>** includes two child Frames: **Frame <02> <01>** and **Frame <02> <02>**.

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

Figure 3.68: Frame <01> with two child Frames before duplication

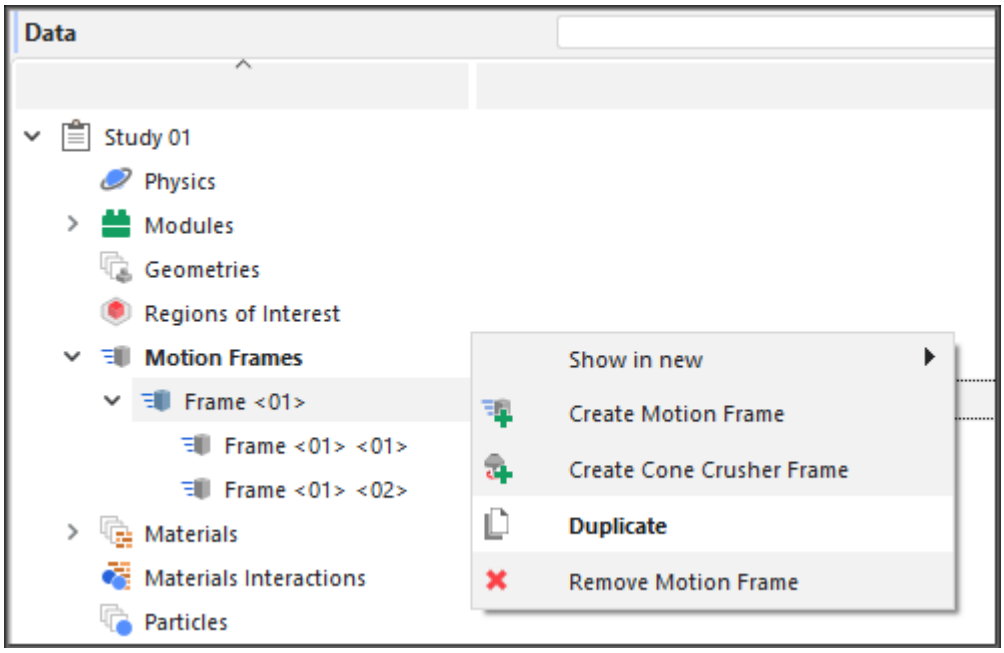
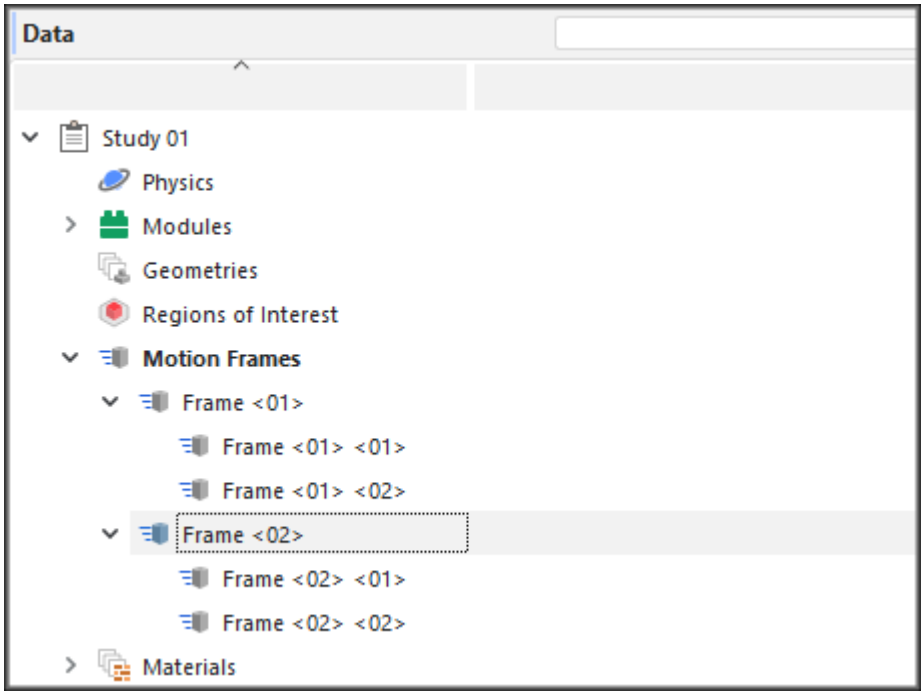


Figure 3.69: After Frame <01> is duplicated, Frame <02> appears with two child Frames



ABOUT MOTION FRAMES WITHIN SIMULATIONS THAT HAVE BEEN COPIED FOR RESTART PURPOSES

Motion Frames that are active at the time a processed simulation is saved for restart purposes (see also Save a Copy of a Partially Processed Simulation for Restart Purposes (p. 855)) will be copied to the new simulation, and will retain their original geometry assignments. However, even though the geometries using the frames can be removed from the copy, the copied frames themselves cannot be removed from the copy. These copied frames will also only allow changes to the **Stop Time** field

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

for the existing motion (as long as the copied motion hasn't completed yet, meaning the **Stop Time** value is positive); new motions can be added to the existing frame, however, by clicking the **Add Motion** button.

In addition, only new motion frames can be assigned to new geometries in the copy; newly added geometries cannot be assigned frames copied from the original simulation, and geometries with motion frames copied from the original simulation cannot be assigned newly added motion frames. (See also About Changing Geometries and/or Particles in a Partially Processed Simulation (p. 847).)

Be aware that if you choose to use parametric expressions in either the **Start Time** or **Stop Time** fields of your original simulation project, know that only the resulting values and not the variables and/or mathematical functions you entered will be retained in the project copy you save for restart purposes. (See also I get a "Links removed" message when I save my project for restart purposes (p. 1381).)

About Motion Frame References

Each Motion Frame has its own orientation reference (i.e., coordinate system) upon which its movements are based. The current (i.e., instantaneous) orientation is represented in the Motion Preview window by the axis for the Frame.

In this version of Rocky, all Frames use only an implicit "local" reference, which uses the current orientation of the selected Frame to define the next movement. In this way, the reference is always moving along with the Frame.

Tip:

In this version of Rocky, it is no longer possible to set a "parent" reference for a frame. However, you can recreate similar "parent" reference behavior by using nested Frames.

Note: When opening a project created in an older version of Rocky, any Frames with references set to "parent" must be modified so that they behave correctly in the "local" reference. See also I get "reference" errors in my Motion Frames when I open an older project (p. 1327).

You are able to assign a Motion Frame to Inlets or Outlets for particles or fluid. To do that you need to add a Motion Frame to the surface that is attached to that Inlet or Outlet (see also About Adding and Editing Inlets and Outlets (p. 708)).

You can also add Motion Frames to a Particle Custom Inlet when Rocky detects that the Particle set (p. 1393) assigned to the Particle Custom Inlet is a multi-element frozen Fiber (p. 1391) one. In these cases, the frozen Segments (p. 1394) of the injected fibers will move according to the definition of the motion frame assigned.

See also Apply a Motion Frame to a Geometry (p. 569). See also Apply a Motion Frame to a Custom Input (p. 571).

About Motion Frames and Inlets/Outlets

You are able to assign a Motion Frame to Inlets or Outlets for particles or fluid. To do that you need to add a Motion Frame to the surface that is attached to that Inlet or Outlet (see also About Adding and Editing Inlets and Outlets (p. 708)).

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

You can also add Motion Frames to a Particle Custom Inlet when Rocky detects that the Particle set (p. 1393) assigned to the Particle Custom Inlet is a multi-element frozen Fiber (p. 1391) one. In these cases, the frozen Segments (p. 1394) of the injected fibers will move according to the definition of the motion frame assigned.

See also Apply a Motion Frame to a Geometry (p. 569). See also Apply a Motion Frame to a Custom Input (p. 571).

About Motion Frames and User Processes

User processes created for post-processing, such as cubes and cylinders, can now use motion frames to move around the domain and extract data while following geometries or particles. (see also Filter Views and Data with User Processes (p. 1174)).

See also Apply a Motion Frame to a User Process (p. 570).

Motion Frame Parameters

See the images and tables below to understand how to create and apply Motion Frames to your imported geometries and Particle Custom Inlets.

Figure 3.70: Motion Frames parameters in the Data Editors panel, and the Motion Preview window

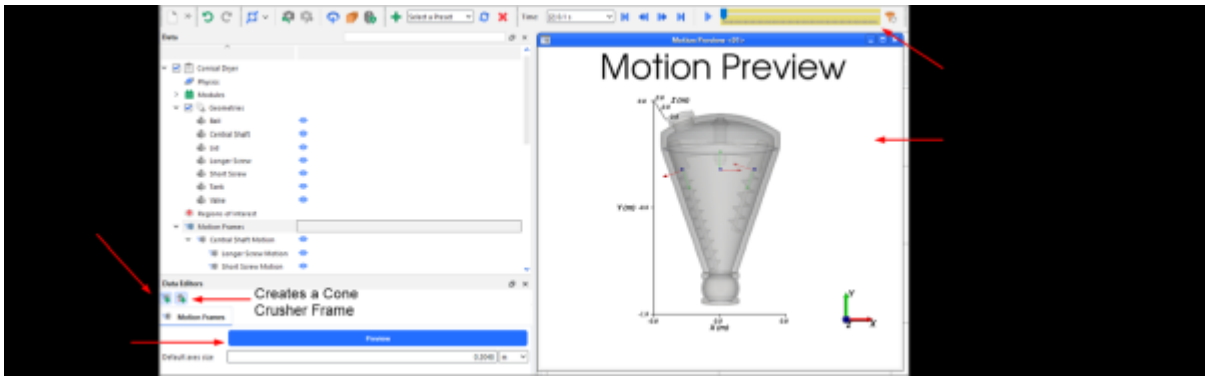
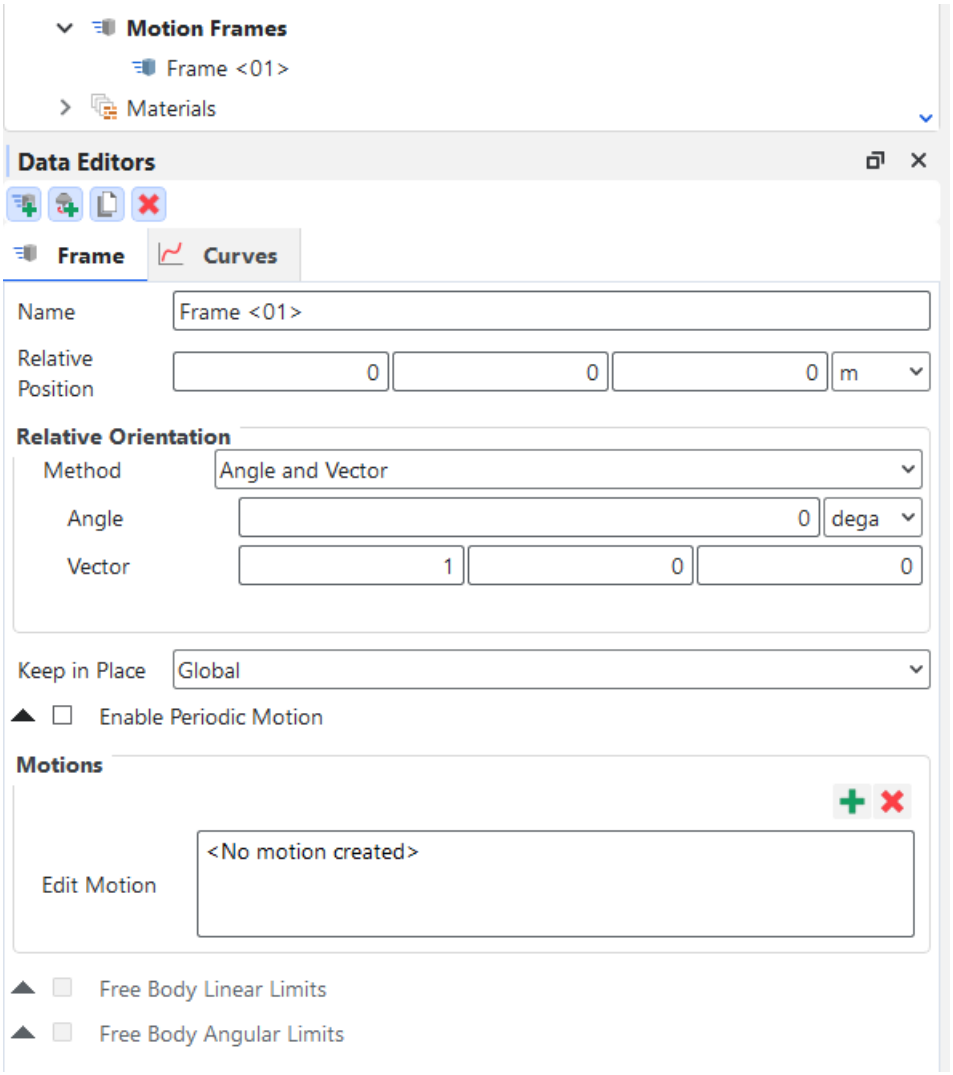


Table 1: Motion Frames parameters (Main Entity)

Setting	Description	Range
Default axes size	Sets the size of the axes used to represent the Motion Frames in the Motion Preview window. Changing the size is useful in cases where the geometries are significantly bigger or smaller than the motion axes, as seeing the axes in relation to the movement of the geometry is critical to understanding and verifying the movement setup. Affects all Motion Frames axes in the project. (See also Preview a Motion in 3D (p. 573).) **Note:** The axes for Motion Frames are different than the axes for the windows themselves; the latter have their own display settings that you can modify. (See also About	Positive value

Setting	Description	Range
	Using the Window Editors Panel to Change the Window Axes Displays (p. 1071).)	

Figure 3.71: Parameters for newly added Frame entry - Angle and Vector



Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

Figure 3.72: Parameters for newly added Frame entry - Basis Vectors

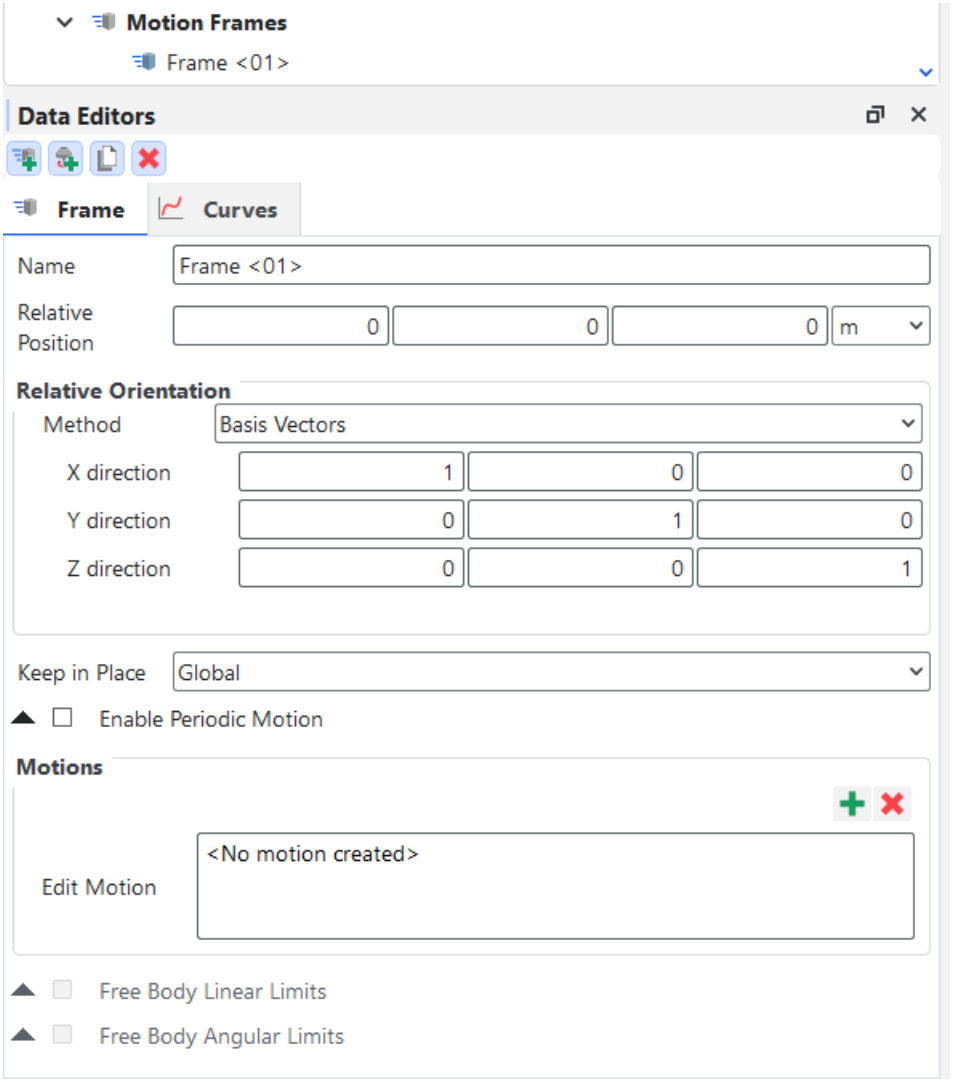
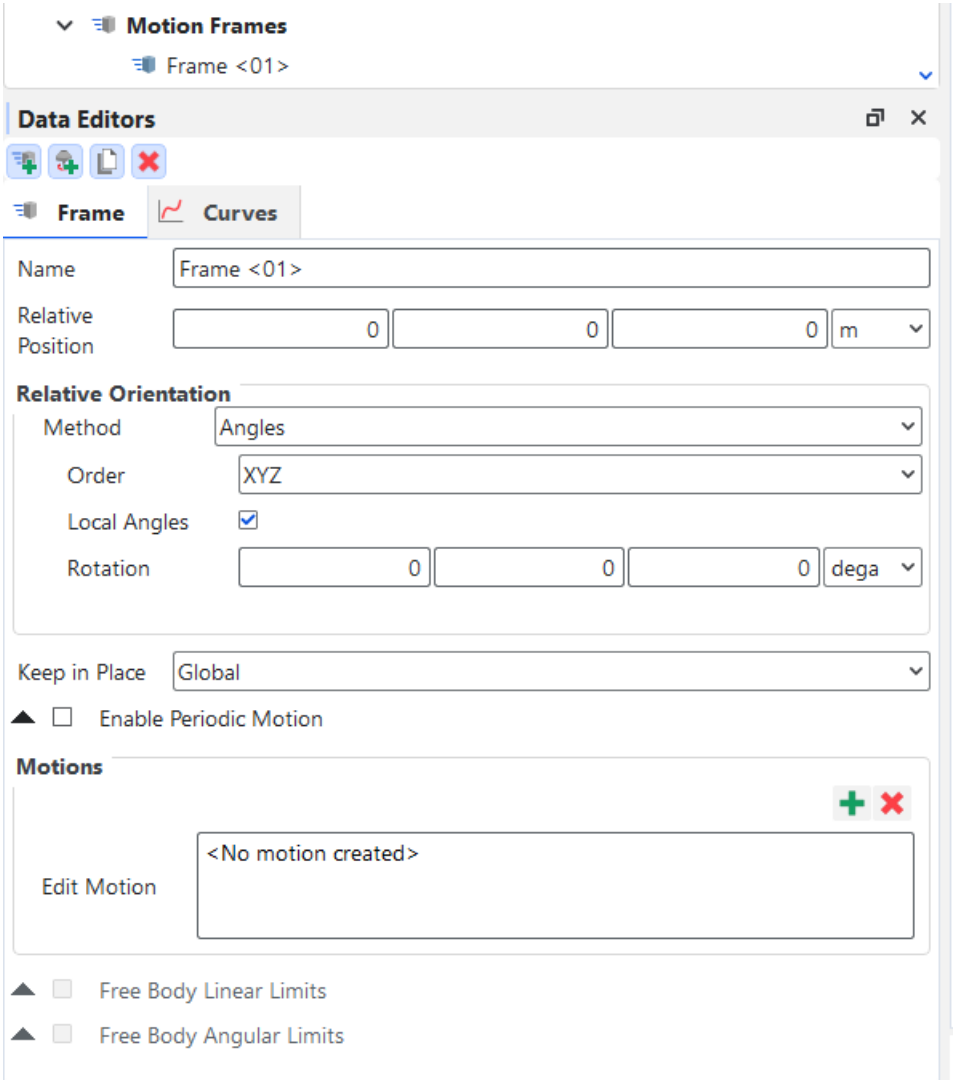


Figure 3.73: Parameters for newly added Frame entry - Angles



Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

Figure 3.74: Motion Frame with Enable Periodic Motion selected

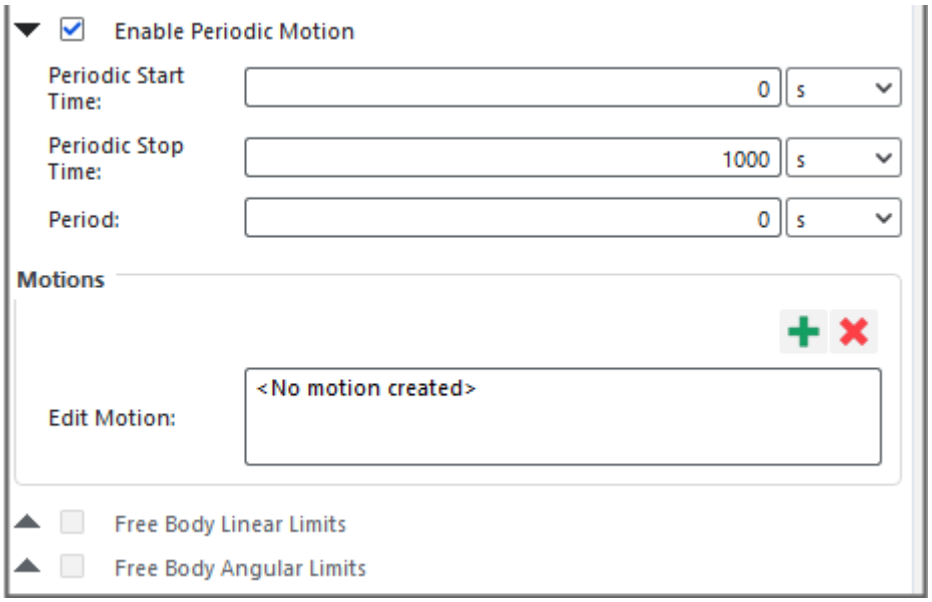
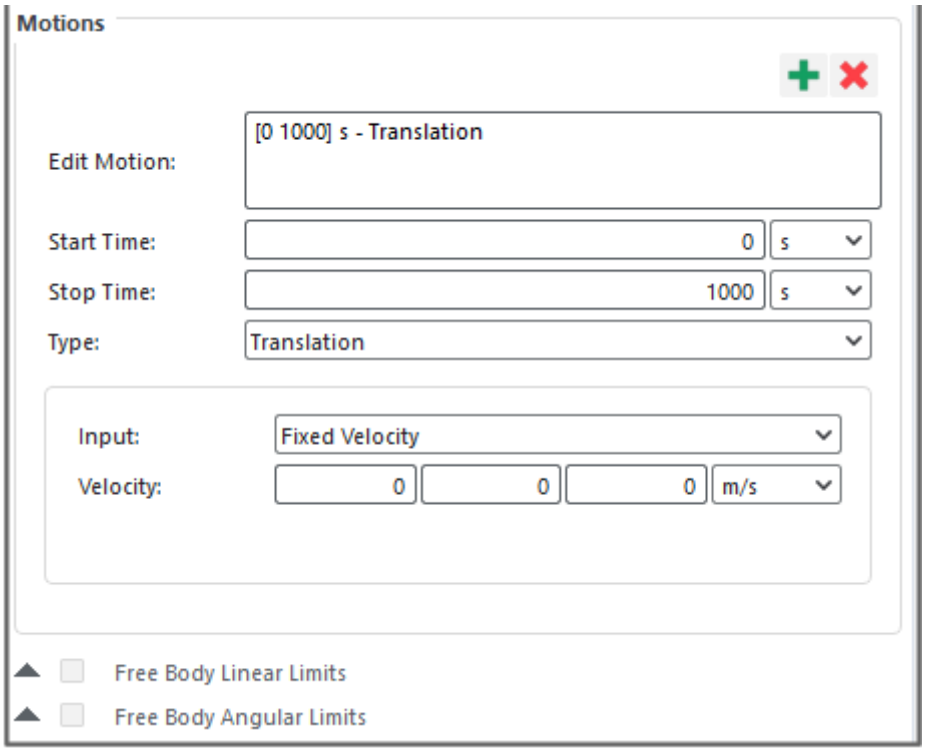


Figure 3.75: New Motion added to Frame entry showing Translation parameters



Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

Figure 3.76: New Motion showing Rotation parameters

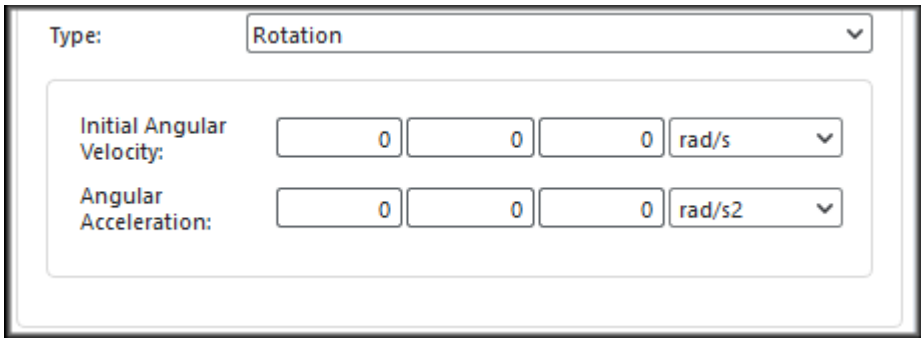


Figure 3.77: New Motion showing Periodic Rotation (Pendulum) parameters

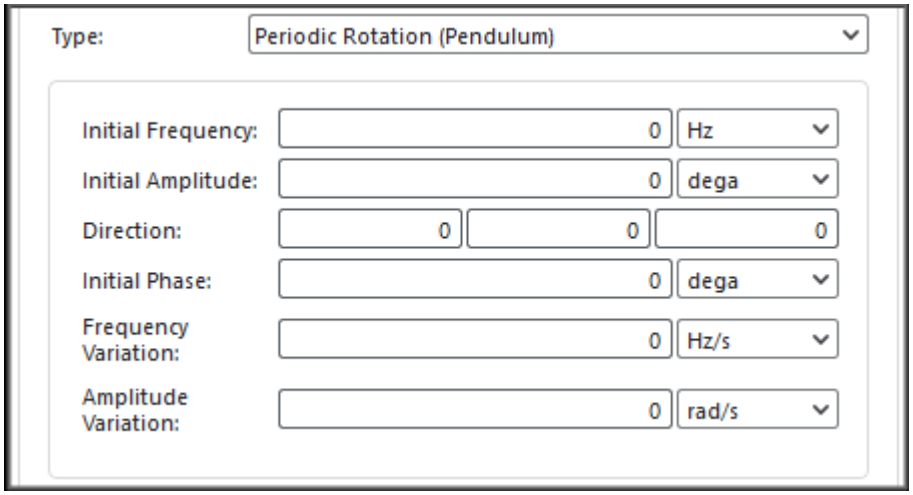
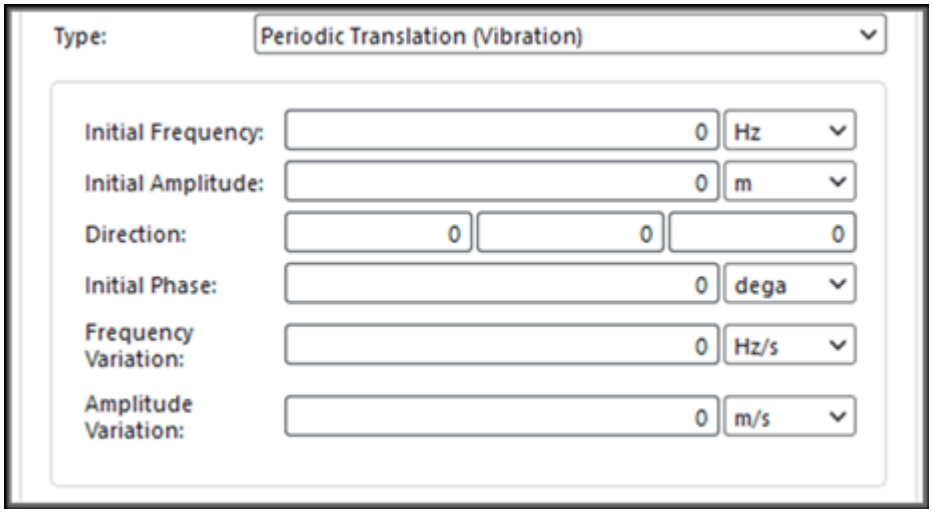


Figure 3.78: New Motion showing Periodic Translation (Vibration) parameters



Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

Figure 3.79: New Motion showing Free Body Translation parameters

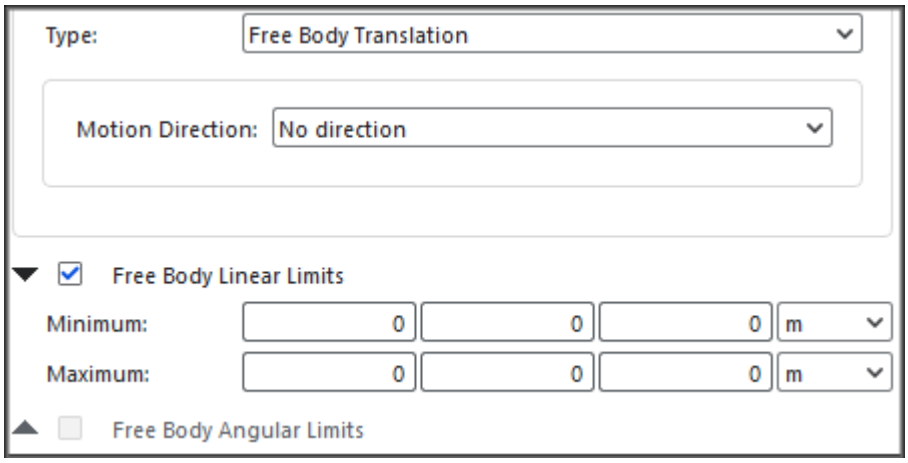


Figure 3.80: New Motion showing Free Body Rotation parameters

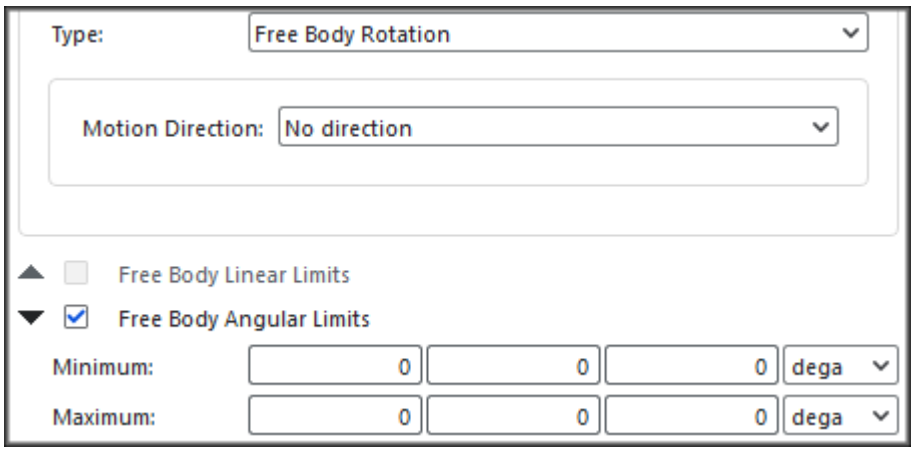


Figure 3.81: New Motion showing Additional Force parameters

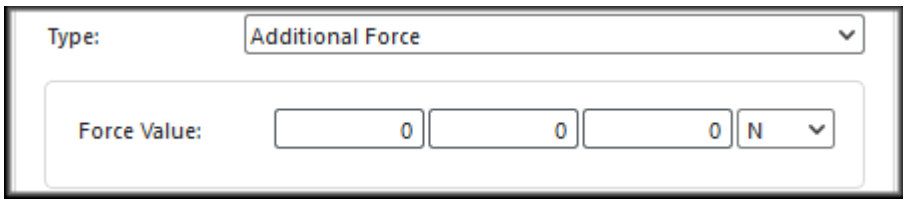
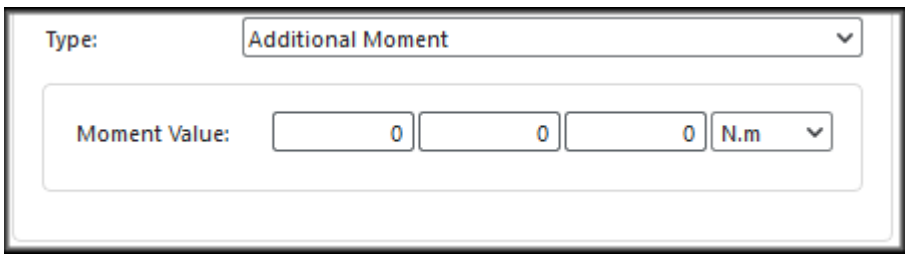


Figure 3.82: New Motion showing Additional Moment parameters



Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

Figure 3.83: New Motion showing Spring-Dashpot Force parameters

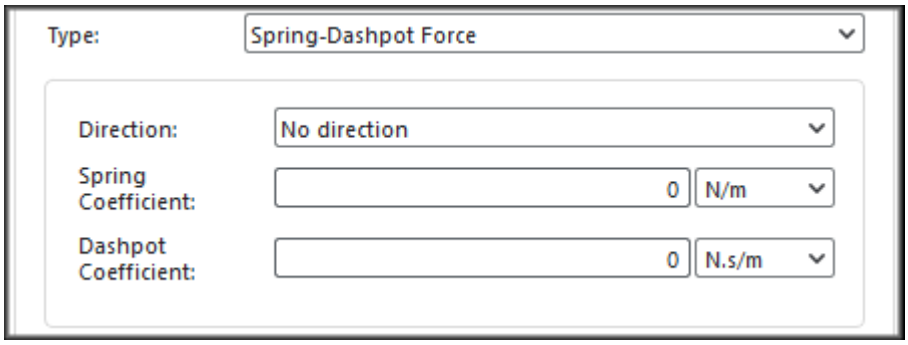


Figure 3.84: New Motion showing Spring-Dashpot Moment parameters

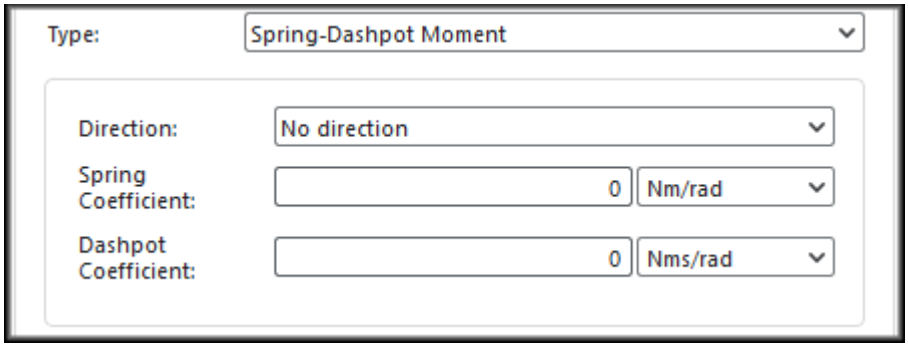


Figure 3.85: New Motion showing Linear Time Variable Force parameters

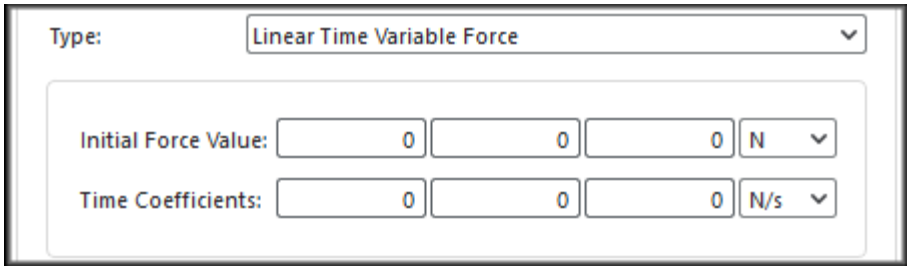
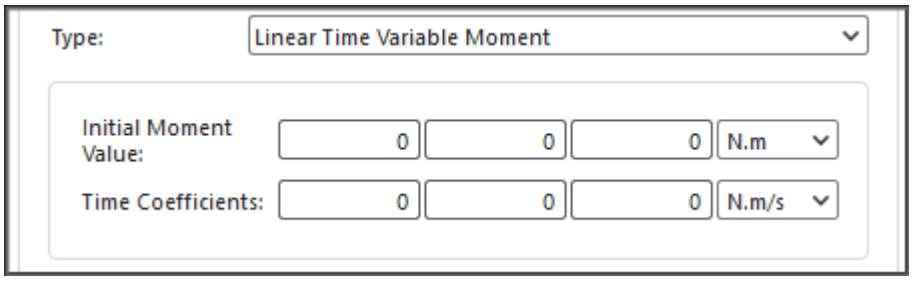


Figure 3.86: New Motion showing Linear Time Variable Moment parameters



Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

Figure 3.87: New Motion showing Time Series Translation parameters

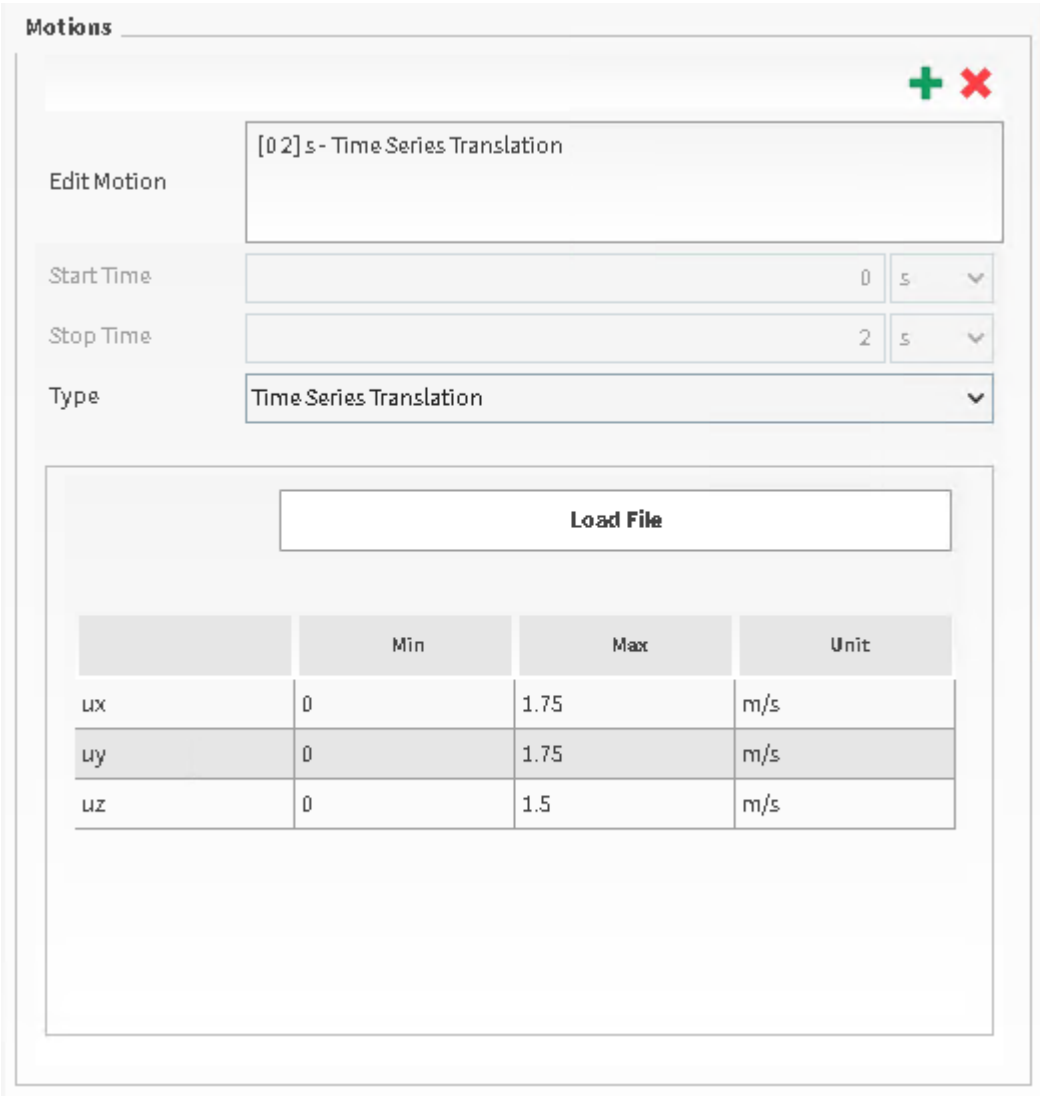


Figure 3.88: New Motion showing Time Series Rotation parameters

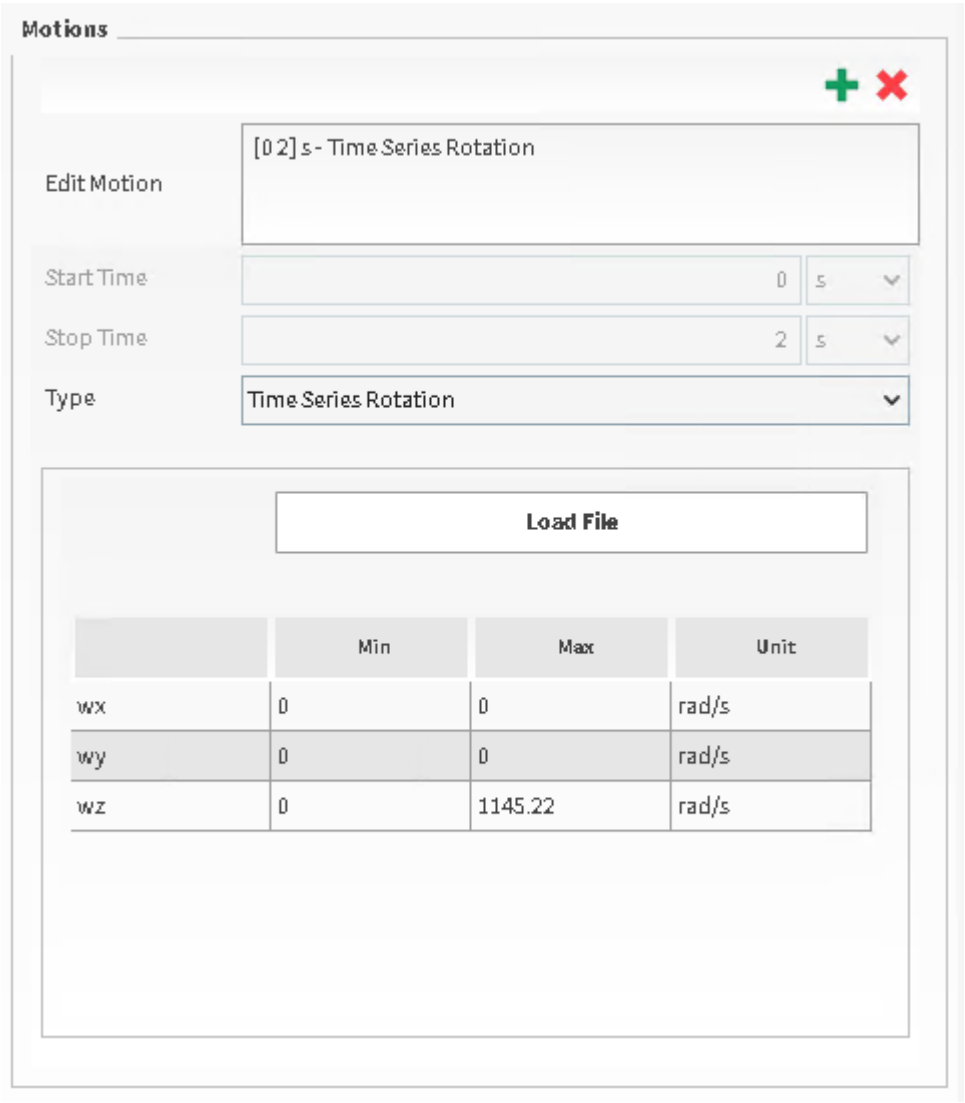


Table 2: Motion Frames parameters (Individual Frames)

Setting	Description	Range
Name	Enables you to set a unique identifier for the selected motion frame. **Note:** When importing movements from Fluent, this will be equal to the **Zone Name** that was set for the Cell Zone in Fluent.	99 character limit

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

Setting	Description	Range
Relative Orientation	Enables you to select how you want to define the orientation of the motion frame shape. Specifically: • **Angles:** enables you to define angles of rotation in three directions, the order of which you can also specify. • **Angle and Vector:** enables you to define one vector and one angle of rotation around it. • **Basis Vectors:** enables you to define the X, Y, and Z directions of the motion frame local basis.	Angles; Angle and Vector; Basis Vectors. Angle and Vector; Basis Vectors.
Relative Orientation Angle and Vector		
Angle	When Angle and Vector is selected for Orientation, this is the angle the motion frame will rotate around the Vector defined.	Any value
Vector	When Angle and Vector is selected for Orientation, this is the X, Y, and Z components that define the vector around which the motion frame will rotate, using the Angle defined.	No limit but the values entered will be normalized
Relative Orientation Basis Vector		
X direction	When Basis Vectors is selected for Orientation, this is the coordinate values defining the first of three directional vectors that together define the final orientation of the motion frame. **Note:** To ensure correct results, make sure you define this vector as orthogonal (perpendicular) to the other two vectors.	No limit but,the values entered will be normalized
Y direction	When Basis Vectors is selected for Orientation, this is the coordinate values defining the second of three directional	No limit but,the values

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

Setting	Description	Range
	vectors that together define the final orientation of the motion frame. **Note:** To ensure correct results, make sure you define this vector as orthogonal (perpendicular) to the other two vectors.	entered will be normalized
Z direction	When Basis Vectors is selected for Orientation, this is the coordinate values defining the third of three directional vectors that together define the final orientation of the motion frame. **Note:** To ensure correct results, make sure you define this vector as orthogonal (perpendicular) to the other two vectors.	No limit but,the values entered will be normalized
Relative Orientation		
Angles		
Order	When Angles is selected for Orientation, this defines the order in which the three Rotation text fields will be applied.	XYZ; ZXY; YXZ; YZX; ZXY; ZYX.
Local Angles	When Angles is selected for Orientation, this defines what coordinate system will be used as a basis for the angle specified. Specifically: • When enabled, the angle will be based on the local coordinate system. • When cleared, the x angle will be based on the global coordinate system.	Turns on or off
Rotation	When Angles is selected for Orientation, this is the degree of cube rotation in each of the three directions specified by the Order provided.	Any value

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

Setting	Description	Range
Keep in Place	Specifies whether or not the motion frame to which the frame is assigned will appear to move physically from one location to another. Specifically: • **Disabled**: Enables the geometry to which the frame is assigned to appear to move physically from one location to another. For example, a gate that rises or a mill that rotates. • **Global**: Enables the geometry to which the frame is assigned to appear stationary but still have surface velocity applied. For example, a stationary conveyor whose belt moves particles. • **Local**: For children frames only, enables the geometry to which the frame is assigned to appear stationary in relation to its parent frame, although it still has its movement. Otherwise (for non - children frames), it does not differ from the **Global** option. Notes: • **Keep in Place** (Global) is how all default feed and receiving conveyors in Rocky are set up. • Any gaps in the movement time periods will be treated as no motion. See also I get "Displacement" errors in my Motion Frames when I open an older project (p. 1333).	Turns on or off
Enable Periodic Motion	Enables the motion(s) defined for the selected frame to be repeated or looped for a defined period within the simulation. (See also Use Motion Frames to Enable Geometry Motions to be Repeated (p. 590).)	Turns on or off
Periodic Start Time	When **Enable Periodic Motion** is selected, this defines the amount of delay after the motion(s) **Start Time** and **Stop Time** to begin and end the motion to be repeated. Specifically: • Setting zero (0) here means the periodic motion will start and stop exactly when the motion(s) are set to start and stop, with no delay. • Setting a value greater than zero will delay the motion(s) from starting by the amount you define. For example, if a motion's **Start Time** is set to 5 s and its **Stop Time** is set to 10 s, and you set this value to 2 s, then the periodic motion(s) will not begin until 7 s and will not end until 12 s into the simulation.	Positive values

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

Setting	Description	Range
Periodic Stop Time	When **Enable Periodic Motion** is selected, this defines the time during the simulation when the motion(s) will stop, regardless of what is set for **Stop Time**. Specifically: • Setting this value equal to or less than the actual start time of your periodic motions (defined here as the **Start Time** value plus the **Periodic Start Time** value) will result in no movements occurring. • Setting this value greater than the actual start time of your periodic motions but by an amount less than the **Period** will result in only part of your periodic motion(s) occurring. • Setting this value greater than the actual start time of your periodic motions and by an exact multiple of the **Period** will result in complete repeats of your periodic motion(s). **Tip:** To ensure that the periodic motion continues for the entire simulation, keep this value set to the default (1000 s) or set it higher than your simulation duration.	Positive values
Period	When **Enable Periodic Motion** is selected, this defines how much of the original motion(s) will be repeated. Specifically: • Entering the exact time amount for which you have specified motion(s) will result in full repeats of the original motion(s). • Entering a smaller time amount will result in partial repeats of the original motions(s). For example, if your original motion was from 0 to 4 s (4 s long), setting this value to 2 results in only the first half of the original motion being repeated. • Entering a larger time amount will result in full repeats followed by a period of no motion. For example, if your original motion was from 0 to 5 s (5 s long), setting this value to 7 results in 2 s of no motion after each 5 s period of motion.	Positive values
Edit Motion	Lists the individual motions you have defined for the selected motion frame. Motions are automatically named in accordance with the following method: **[StartTime StopTime]s - Type**	Automatically determined

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

Setting	Description	Range
Start Time	The time you want the selected motion to begin. Note:*If you choose to use parametric expressions in this field, know that only the resulting value and not the variables and/or mathematical functions you enter will be retained in any project copies you save for restart purposes.* (See also I get a "Links removed" message when I save my project for restart purposes (p. 1381).)	Positive values **Tip:** Tip: Check the Status panel to ensure that any variables or mathematical functions you might use results in valid values. (See also Double-Click the Status Panel to Jump to the Appropriate UI Location (p. 117).)
Stop Time	The time you want the selected motion to end. **Note:** Any gaps of time between multiple motions will be interpreted as no movement. Note: *If you choose to use parametric expressions in this field, know that only the resulting value and not the variables and/or*	Positive values **Tip:** Check the **Status** panel

Setting	Description	Range
	mathematical functions you enter will be retained in any project copies you save for restart purposes. (See also I get a "Links removed" message when I save my project for restart purposes (p. 1381).)	to ensure that any variables or mathematical functions you might use results in valid values. (See also Double-Click the Status Panel to Jump to the Appropriate UI Location (p. 117).)
Type	Defines the type of movement. Specifically: • **Translation**: Moves the frame in a given direction. (Linear continuous motion.) • **Rotation**: Rotates the frame about its center point over a given direction. (Angular continuous motion.) • **Periodic Rotation (Pendulum)**: Rotates the frame periodically about its center point over a given direction with specified frequency and amplitude. (Angular oscillatory motion.) • **Periodic Translation (Vibration)**: Displaces the frame periodically in a given direction with specified frequency and amplitude. (Linear oscillatory motion.) • **Free Body Translation**: Allows the frame to freely translate in a given direction due to forces-such as the ones generated by gravity, particle interactions, or	Translation; Rotation; Periodic Rotation (Pendulum); Periodic Translation (Vibration); Free Body Translation; Free Body Rotation; Additional Force; Additional Moment; Spring-Dashpot Force; Spring-Dashpot Moment;

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

Setting	Description	Range
	(separately set) **Additional Force**, **Spring-Dashpot Force**, and/or **Linear Time Variable Force** motions- acting upon the frame. • **Free Body Rotation** : Allows the frame to freely rotate about its center point over a given direction due to forces-such as the ones generated by gravity, particle interactions, or (separately set) **Additional Moment** , **Spring-Dashpot Moment**, and/or **Linear Time Variable Moment** motions- acting upon the frame. • **Time Series Translation** : Displays the maximum, minimum, and unit (m/s) for the columns (ux, uy, uz). The accepted formats for input files are **.csv and .xlsx.** • **Time Series Rotation** : Displays the maximum, minimum, and unit (rad/s) for the columns (wx, wy, wz).The accepted formats for input files are **.csv and .xlsx.** • **Additional Force**: When used in conjunction with **Free Body Translation**, this defines an additional (prescribed) force to act upon the frame. Unlike standard translation motions, which prescribe a fixed velocity and/or acceleration, this additional force is included in the force balance as well as gravity and particle interactions, which results in a velocity that adapts to its conditions. • **Additional Moment**: When used in conjunction with **Free Body Rotation**, defines an additional (prescribed) moment (torque) to act upon the frame. Unlike standard rotation motions, which prescribe a fixed angular velocity and/or acceleration, this additional torque is included in the moment balance as well as moments from gravity and particle interactions, which results in an angular velocity that adapts to its conditions. • **Spring-Dashpot Force**: When used in conjunction with **Free Body Translation**, this provides a combination of two forces-the spring force and the dashpot (damper) force- that can be used to model dampers or friction between motion frames that are translating. The spring force increases as the frame moves away from its original position, trying to bring the frame back to that position. The equation to calculate this force is: $F_s = -kd$ where: F_s is the spring force, k is the spring coefficient, and d is the distance of the frame to its original position. The dashpot (damper) force increases as the frame moves faster (just like a damper, the faster you move, the higher the damper resists), trying to bring the frame back to the zero velocity. The equation to calculate this force is: $F_d = -cv$ where: F_d	Linear Time Variable Force; Linear Time Variable Moment

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

Setting	Description	Range
	is the dashpot force, c is the dashpot coefficient, and v is the velocity of the frame. • **Spring-Dashpot Moment**: When used in conjunction with **Free Body Rotation**, this provides a combination of two moments-the spring moment and the dashpot (damper) moment-that can be used to model dampers or friction between motion frames that are rotating. The spring moment tries to return the frame to its original orientation (instead of the original position). The equation is: $M_s=k_m\theta$ where: M_s is the spring moment, k_m is the spring (angular) coefficient, and θ is the angle that the frame has being rotated. The dashpot moment tries to stop the rotation of the frame (instead of the translation). The equation is: $M_d=-c_m\omega$ where: M_d is the dashpot moment, c_m is the dashpot (angular) coefficient, and ω is the angular velocity of the frame. • **Linear Time Variable Force**: When used in conjunction with **Free Body Translation**, this applies to the frame a force that either continuously increases or continuously decreases in a linear way over time. The actual value of the force applied over the frame, f, is calculated by adding the initial value of the force, f_0, to the product of the force time coefficients, c_f and the motion time, Δt_f, which can be expressed as: $\mathrm{f}=\mathrm{f}_0+c_f\Delta t_f$ This force can be used to model sample compression tests with linearly increasing compression loads, for example. • **Linear Time Variable Moment**: When used in conjunction with **Free Body Rotation**, this applies to the frame a moment (torque) that either continuously increases or continuously decreases in a linear way over time. The actual value of the moment applied over the frame, m, is calculated by adding the initial value of the moment, m_0, to the product of the moment time coefficients, c_m, and the motion time, Δt_m, which can be expressed as: $m=m_0+c_m\Delta t_m$	
Translation Parameters		
Input	Determines what velocity and acceleration values you want the motion to consider. Specifically: • **Fixed Velocity** enables you to set only a fixed velocity with no acceleration.	Fixed Velocity; Initial and Final Velocity;

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

Setting	Description	Range
	• **Initial and Final Velocity** enables you to set an initial and final velocity and then have Rocky compute the acceleration required to achieve those values. • **Initial Velocity and Acceleration** enables you to set both the initial velocity and acceleration and then have Rocky compute the final velocity that results from those values.	Initial Velocity and Acceleration
Velocity	When **Fixed Velocity** is chosen for **Input** , this enables you to set a single translational velocity in the X, Y, and Z directions respectively.	Any value
Initial Velocity	When **Initial and Final Velocity** or **Initial Velocity and Acceleration** is chosen for **Input**, this sets the starting translational velocity of the selected motion as defined in the X, Y, and Z directions respectively. A positive or negative value will prescribe the orientation of the movement over a given direction.	Any value
Final Velocity	When **Initial and Final Velocity** is chosen for **Input**, this sets the ending translational velocity of the selected motion as defined in the X, Y, and Z directions respectively. A positive or negative value will prescribe the orientation of the movement over a given direction.	Any value
Acceleration (computed)	When **Initial and Final Velocity** is chosen for **Input**, this displays the amount of acceleration in the X, Y, and Z directions respectively that Rocky calculates is required to achieve the **Final Velocity** given the **Initial Velocity** value.	Display only; values are calculated by Rocky
Acceleration	When **Initial Velocity and Acceleration** is chosen for **Input**, this sets the amount of acceleration of the selected motion as defined in the X, Y, and Z directions respectively.	Any value
Final Velocity (computed)	When **Initial Velocity and Acceleration** is chosen for **Input**, this displays the final velocity in the X, Y, Z directions respectively that Rocky calculates given both the **Initial Velocity** and **Acceleration** values.	Display only; values are calculated by Rocky
Rotation Parameters		
Initial Angular Velocity	Sets the starting rotational (angular) velocity of the selected motion as defined in the X, Y, and Z directions respectively. • **Positive values** rotate the frame counter-clockwise around its axis • **Negative values** rotate the frame clockwise around its axis	Any value
Angular Acceleration	Sets the amount of rotational (angular) acceleration of the selected motion as defined in the X, Y, and Z directions respectively.	Any value

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

Setting	Description	Range
Pendulum and Vibration Parameters		
Initial Frequency	When **Pendulum** or **Vibration** is chosen for **Type**, this sets the starting frequency of the of the selected motion.	Any value
Initial Amplitude	When **Pendulum** or **Vibration** is chosen for **Type**, this sets the starting amplitude of the selected motion.	Any value
Direction	When **Pendulum** or **Vibration** is chosen for **Type**, this is the X, Y, and Z vector components that define the direction of the selected motion. Specifically: • For **Pendulums**, this vector defines the 2D plane upon which the swinging motion will occur. • For **Vibrations**, this vector defines the direction over which the vibrating motion will occur.	No limit but values entered will be normalized
Initial Phase	When **Pendulum** or **Vibration** is chosen for **Type**, this determines the angular degree at which the motion begins along the sine wave that defines the oscillating movement. Specifically: • **Zero degrees** (default): The motion begins at the center of the sine wave (Figure 1). • **90 degrees**: The motion begins at the highest (maximum amplitude) limit location along the sine wave (Figure 2). • **270 degrees**: The motion begins at the lowest (minimum amplitude) limit location along the sine wave.	Any value
Frequency Variation	When **Pendulum** or **Vibration** is chosen for **Type**, this determines the amount of variation in frequency per unit of time for the selected motion starting from the **Initial Frequency** value.	Any value
Amplitude Variation	When **Pendulum** or **Vibration** is chosen for **Type**, this determines the amount of variation in amplitude per unit of time for the selected motion starting from the **Initial Amplitude** value.	Any value
Free Body Translation and Free Body Rotation Parameters (Otherwise known as		

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

Setting	Description	Range
Six Degrees of Freedom (6DOF))		
Free Motion Direction	When **Free Body Translation** or **Free Body Rotation** are chosen for **Type**, this determines the axis or axes that free motion is allowed. Specifically: • **No direction**: Free Motion is disabled. • **X direction**: Free Motion is enabled along the X axis only. • **Y direction**: Free Motion is enabled along the Y axis only. • **Z direction**: Free Motion is enabled along the Z axis only. • **X and Y directions** : Free Motion is enabled along the X and Y axes only. • **X and Z directions** : Free Motion is enabled along the X and Z axes only. • **Y and Z directions** : Free Motion is enabled along the Y and Z axes only. • **All directions**: Free Motion is enabled along all axes.	No direction; X direction; Y direction; Z direction; X and Y directions; X and Z directions; Y and Z directions; All directions
Free Body Limits		
Free Body Linear Limits	When **Free Body Translation** is selected from the **Type** list, this enables you to limit the linear movement to occur only between the **Minimum** and **Maximum** coordinate locations you set.	Turns on or off
Free Body Angular Limits	When **Free Body Rotation** is selected from the **Type** list, this enables you to limit the angular movement to occur only between the **Minimum** and **Maximum** coordinate locations you set.	Turns on or off
Minimum	When **Free Body Linear Limits** or **Free Body Angular Limits** are enabled, this sets the location of the minimum limit through which you want free body movements allowed, as defined in the X, Y, and Z directions respectively.	Any value
Maximum	When **Free Body Linear Limits** or **Free Body Angular Limits** are enabled, this sets the location of the maximum limit through which you want free body movements allowed, as defined in the X, Y, and Z directions respectively.	Any value
Additional Force		

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

Setting	Description	Range
and Additional Moment Parameters		
Force Value	When *Additional Force* is chosen for **Type** , this enables you to set the amount of additional, applied force you want acted upon the selected frame. This value is defined in the X, Y, and Z axes respectively, and is itself affected by particle interactions and gravity. **Note:** This motion is designed to only be used in tandem with a **Free Body Translation** motion.	Any value
Moment Value	When *Additional Moment* is chosen for **Type** , this enables you to set the amount of additional, applied moment (torque) you want acted upon the selected frame. This value is defined in the X, Y, and Z axes respectively, and is itself affected by particle interactions and gravity. **Note:** This motion is designed to only be used in tandem with a **Free Body Rotation** motion.	Any value
Spring-Dashpot Force and Spring-Dashpot Moment Parameters		
Spring Coefficient	Defines the stiffness of the spring that attaches the frame to its original position. **Note:** This motion is designed to only be used in tandem with a **Free Body Translation** or **Free Body Rotation** motion.	Any value

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

Setting	Description	Range
Dashpot Coefficient	This value when multiplied by the frame's translational or angular velocity, gives you the actual resistance force or moment. **Note:** This motion is designed to only be used in tandem with a **Free Body Translation** or **Free Body Rotation** motion.	Any value
Linear Time Variable Force and Linear Time Variable Moment Parameters		
Initial Force Value	When **Linear Time Variable Force** is chosen for **Type**, this defines the initial value of the force when the motion begins at its **Start Time**. (See **Type** definitions above for full equations.) and gravity. **Note:** This motion is designed to only be used in tandem with a **Free Body Translation** motion.	Any value
Initial Moment Value	When **Linear Time Variable Moment** is chosen for **Type**, this defines the initial value of the moment (torque) when the motion begins at its **Start Time**. (See **Type** definitions above for full equations.) **Note:** This motion is designed to only be used in tandem with a **Free Body Rotation** motion.	Any value
Time Coefficients	When either **Linear Time Variable Force** or **Linear Time Variable Moment** are selected as **Type**, this coefficient is used to define the actual force (or moment) that will be applied to the motion frame by multiplying it by the motion time to obtain the load as a function of time, which will then be added to the **Initial Force Value** (or **Initial Moment**	Any value

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

Setting	Description	Range
	Value) defined. (See **Type** definitions above for full equations.) **Note:** This motion is designed to only be used in tandem with a **Free Body Translation** or **Free Body Rotation** motion.	
External Velocity Profile		
Time Series Translation	When **Time Series Translation** is chosen for **Type**, this defines the movement displayed in the maximum, minimum, and unit (m/s) for the columns: • Time • X component of linear velocity (ux) • Y component of linear velocity (uy) • Z component of linear velocity (uz)	Positive Values
Time Series Rotation	When **Time Series Rotation** is chosen for **Type**, this defines the movement displayed in the maximum, minimum, and unit (m/s) for the columns: • Time • X component of angular velocity (wx) • Y component of angular velocity (wy) • Z component of angular velocity (wz)	Positive Values

What would you like to do?

- Create a Motion Frame (p. 569)
- Apply a Motion Frame to an Imported Geometry (p. 569)
- Apply a Motion Frame to a Custom Input (p. 571)
- Preview a Motion in 3D (p. 573)
- Create and Modify a Motion Preview Window (p. 571)

CREATE AND APPLY CONE CRUSHER FRAMES

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

A Cone Crusher Frame is a specific type of Motion Frame that based upon the parameters you define, automatically creates two separate Frames that you can apply to your mantle and shaft geometries. Because the Frames use free body motions, a Cone Crusher Frame cannot yet be previewed in the same manner as a Motion Frame.

What would you like to do?

- Learn more About Creating and Applying Cone Crusher Frames to Imported Geometries (p. 565)
- Create a Cone Crusher Frame (p. 567)
- Apply a Cone Crusher Frame to Custom Mantle and Shaft Geometries (p. 568)
- Use a Cone Crusher Frame to Enable a Geometry to Adopt a Crushing Motion (p. 589)

See Also:

- About Importing Geometry Components (p. 487)
- Create and Apply Motion Frames (p. 532)

About Creating and Applying Cone Crusher Frames to Imported Geometries

A Cone Crusher Frame is a unique kind of Motion Frame that simulates a crushing movement. It does so by enabling the frame to be inclined at a certain angle while being rotated around a vertical axis. An example of this kind of movement is when an inclined mantle is made to rotate around the vertical shaft of a cone crusher. (See also Crusher Simulation.)

Although the parameters for a Cone Crusher Frame are unique to this particular kind of frame, it is applied to your imported geometries in a manner similar to a standard Motion Frame. The main difference is that when you go to apply your Cone Crusher Frame to your crusher geometries, two separate Motion Frames will be listed in the **Motion Frame** box: one for your mantle geometry, and one for your shaft geometry. Rocky automatically creates these two frames for you based upon the parameters you define for the Cone Crusher Frame. Also, due to the Free Body Motion movements included, you cannot currently preview these frames in the Motion Preview window.

Tip:

To see a walk-through example of setting up and applying a Cone Crusher Motion Frame, refer to the following Workshop: Tutorial - Cone Crusher in the *Rocky Tutorial Guide*.

See the images and table below to understand how to create and apply a Cone Crusher Frame to your imported geometries.

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

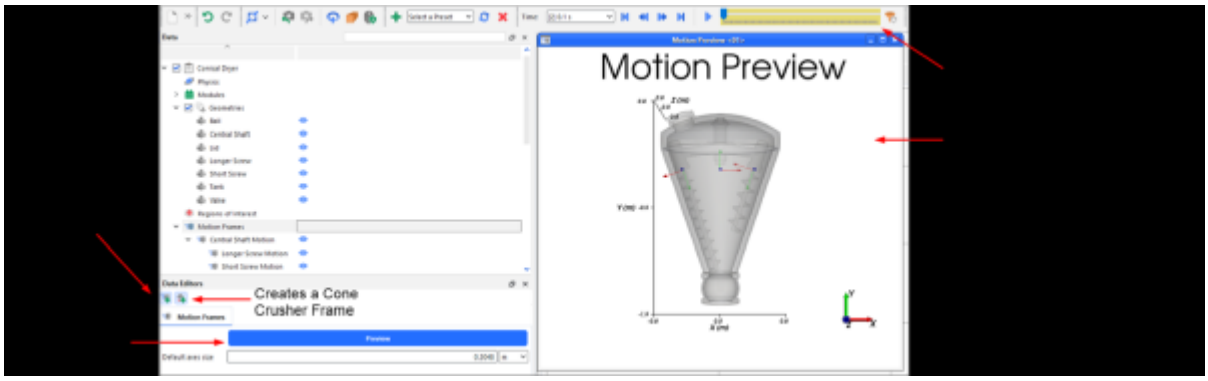


Figure 1: Motion Frames parameters in the Data Editors panel, and the Motion Preview window * Motion Preview window works for all motions except for free body motions, which includes the ones created by the Cone Crusher Frame

Figure 3.89: Parameters for newly added Cone Crusher Frame

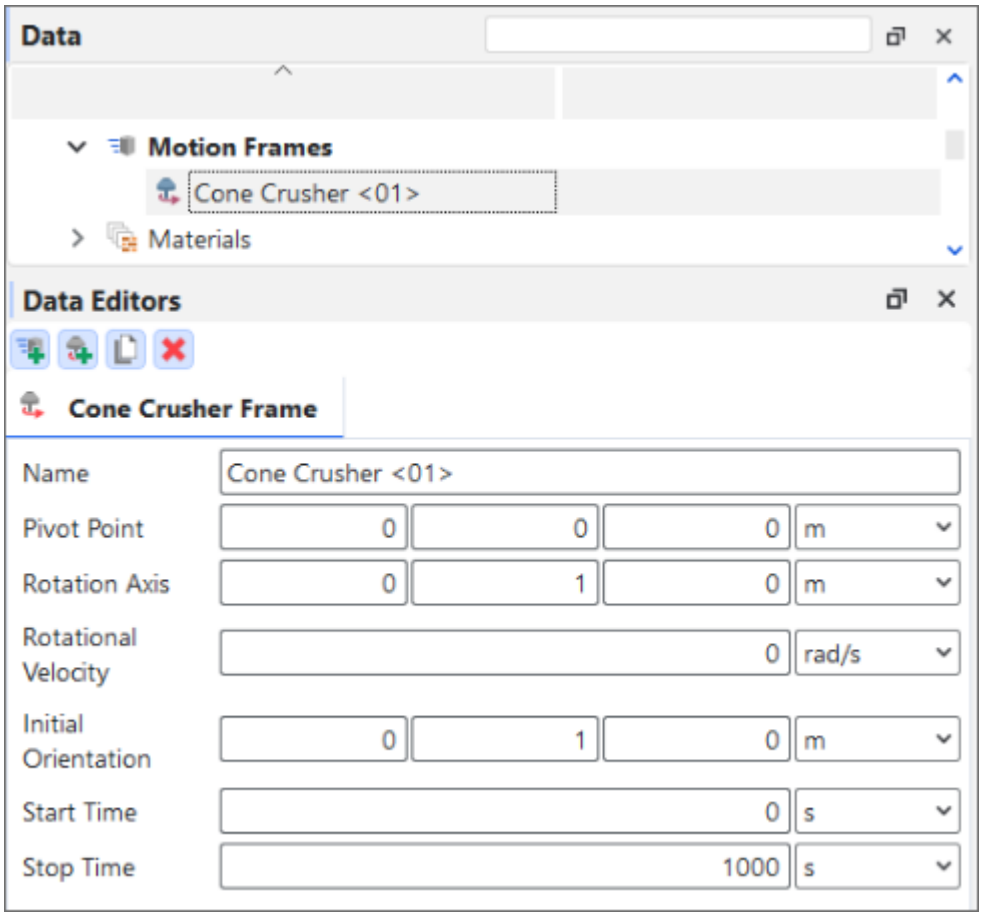


Table 1: Cone Crusher Frame parameters

Setting	Description	Range
Name	Enables you to set a unique identifier for the selected frame.	99 character limit
Pivot Point	The coordinate location of the point around which the crusher and shaft will pivot, as specified in the	No limit

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

Setting	Description	Range
	X, Y, and Z directions. For first-level frames, this is based upon the global coordinate system. For nested or child frames, this is based upon the local coordinate system of the parent frame.	
Rotation Axis	The X, Y, and Z components that define the vector about which the crusher shaft and mantle are rotated. For first-level frames, this is based upon the global coordinate system. For nested or child frames, this is based upon the local coordinate system of the parent frame.	No limit but the values entered will be normalized
Rotational Velocity	The angular velocity along the **Rotation Axis**.	No limit
Initial Orientation	The X, Y, and Z components that define the vector about which the crusher shaft and mantle begin their rotation.	No limit but the values entered will be normalized
Start Time	The time you want the crushing motion to begin.	Positive values
Stop Time	The time you want the crushing motion to end.	Positive values

What would you like to do?

- Create a Cone Crusher Frame (p. 567)
- Apply a Cone Crusher Frame to Custom Mantle and Shaft Geometries (p. 568)
- Use a Cone Crusher Frame to Enable a Geometry to Adopt a Crushing Motion (p. 589)

Create a Cone Crusher Frame

1. From the **Data** panel, select **Motion Frames**.
2. From the **Data Editors** panel, ensure **Default axes size** is set the way you want, and then do one of the following:
   - To create a new parent Cone Crusher Frame, click the **Create Cone Crusher Frame** button. A new Cone Crusher entry appears under **Motion Frames** in the **Data** panel.
   - To create a new nested (child) Cone Crusher Frame, from the **Data** panel, select the parent Frame entry to which you want to add a child Cone Crusher Frame, and then from the **Data Editors** panel, click the **Create Cone Crusher Frame** button. A new Cone Crusher entry appears under the parent Frame you selected in the **Data** panel.

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

3. From the **Data** panel, select the Cone Crusher entry you just added and then from the **Data Editors** panel, enter the values you want. (See also About Creating and Applying Cone Crusher Frames to Imported Geometries (p. 565).)

Tip:

After creating a new Cone Crusher Frame, you must still apply the frame to your mantle and shaft geometries to have the geometries themselves move. (See also Apply a Cone Crusher Frame to Custom Mantle and Shaft Geometries (p. 568).)

Note:

Due to the free body motions involved, you cannot currently use the Motion Preview window to see and test the Cone Crusher Frames in 3D. (See also Preview a Motion in 3D (p. 573).)

See Also:

- About Creating and Applying Cone Crusher Frames to Imported Geometries (p. 565)
- Use a Cone Crusher Frame to Enable a Geometry to Adopt a Crushing Motion (p. 589)

Apply a Cone Crusher Frame to Custom Mantle and Shaft Geometries

1. Ensure the two geometries (mantle and shaft) to which you want to apply the Cone Crusher Frame have been imported.
2. Ensure the Cone Crusher Frame you want to apply has been created. (See also Create a Cone Crusher Frame (p. 567).)
3. From the **Data** panel, select the imported mantle geometry to which you want to apply the Cone Crusher Frame.
4. From the **Data Editors** panel, select the **Geometry** tab, and then from the **Motion Frame** list, select the Cone Crusher Frame you want. (The **Name** will be appended with "Mantle".)
5. From the **Data** panel, select the imported shaft geometry to which you want to apply the Cone Crusher Frame.
6. From the **Data Editors** panel, select the **Geometry** tab, and then from the **Motion Frame** list, select the Cone Crusher Frame you want. (The **Name** will be appended with "Shaft".)

Note:

Due to the free body motions involved, the Motion Preview window cannot currently preview these kinds of frames in 3D. (See also Preview a Motion in 3D (p. 573).)

See Also:

- About Creating and Applying Cone Crusher Frames to Imported Geometries (p. 565)

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

- Use a Cone Crusher Frame to Enable a Geometry to Adopt a Crushing Motion (p. 589)

CREATE A MOTION FRAME

1. From the **Data** panel, select **Motion Frames**.
2. From the **Data Editors** panel, ensure **Default axes size** is set the way you want, and then do one of the following:
   - To create a new parent Motion Frame, click the **Create Motion Frame** button. A new Frame entry appears under **Motion Frames** in the **Data** panel.
   - To create a new nested (child) Motion Frame, from the **Data** panel, select the parent Frame entry to which you want to add a child Motion Frame, and then from the **Data Editors** panel, click the **Create Motion Frame** button. A new Frame entry appears under the parent Frame you selected in the **Data** panel.
3. From the **Data** panel, select the Frame you just added and then from the **Data Editors** panel, do all of the following:
   a. Enter the **Name**, **Relative Position**, **Relative Rotation Vector**, **Rotation Angle**, **Keep in Place**, and **Periodic Motion** values you want. (See also About Creating and Applying Motion Frames (p. 533).)
   b. Under **Motions**, click the **Add Motion** button, and enter the motion values you want.
   c. Repeat step 3b for every motion you want included within the new Motion Frame.

Important:

It is necessary to check/adjust the **Relative Position** parameter of the Motion Frame, and also the **Mass** properties of the Geometry. This is required so the solver can correctly calculate Forces and Momentes the geometry experiences regarding its Center of Mass and the Motion Frame's center.

Tips:

- After creating a new Motion Frame, you must still apply the Motion Frame to an imported geometry or Custom Input to have the component itself move. (See also Apply a Motion Frame to an Imported Geometry (p. 569) or Apply a Motion Frame to a Custom Input (p. 571).)
- After you have applied the Motion Frame to a geometry or Custom Input, you can use the Motion Preview window to see and test the movement in 3D. (See also Preview a Motion in 3D (p. 573).)

See Also:

- About Creating and Applying Motion Frames (p. 533)

APPLY A MOTION FRAME TO A GEOMETRY

You can add Motion Frame to any geometry, wall or surface, created or imported into Rocky. An useful application is to add a Motion Frame to a surface that is attached to an Inlet or Outlet (See

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

also About Adding and Editing Inlets and Outlets (p. 708).) which allows for this inlet/outlet to be moved according to the Motion Frame. To apply a Motion Frame to a geometry do all of the following:

1. Ensure the Motion Frame you want to apply has been created. (See also Create a Motion Frame (p. 569).)
2. From the **Data** panel, select the geometry to which you want to apply a Motion Frame.
3. From the **Data Editors** panel, select the **Wall/Surface** tab, and then from the **Motion Frame** list, select the Motion Frame you want.

Important:

It is necessary to check/adjust the **Relative Position** parameter of the Motion Frame, and also the **Mass** properties of the Geometry. This is required so the solver can correctly calculate Forces and Momentes the geometry experiences regarding its Center of Mass and the Motion Frame's center.

Tips:

- Use the Motion Preview window to see and test the movement in 3D. (See also Preview a Motion in 3D (p. 573).)
- To apply more than one movement to a single geometry, add multiple motions to a single Motion Frame, or create a nested (child) Motion Frame and then apply the child frame to the geometry. (See also Create a Motion Frame (p. 569).)

See Also:

- About Creating and Applying Motion Frames (p. 533)

APPLY A MOTION FRAME TO A USER PROCESS

User processes created for post-processing, such as polyhedron (envelope), cubes and cylinders, can use motion frames to move around the domain and extract data while following geometries or particles.

1. Ensure the Motion Frame you want to apply has been created. (See also Create a Motion Frame (p. 569).)
2. From the **Data** panel, select the User Process to which you want to apply a Motion Frame. (See also Filter Views and Data with User Processes (p. 1174).)
3. From the **Data Editors** panel, select the **Polyhedron (Envelope)/Cube/Cylinder** tab, and then from the **Motion Frame** list, select the Motion Frame you want.

Tip:

To apply more than one movement to a single User Process, add multiple motions to a single Motion Frame, or create a nested (child) Motion Frame and then apply the child frame to the User Process. (See also Create a Motion Frame (p. 569).)

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

See Also:

- About Creating and Applying Motion Frames (p. 533)

APPLY A MOTION FRAME TO A PARTICLE CUSTOM INLET

1. Ensure that the Particle Custom Inlet to which you want to apply a Motion Frame has been created. If the **Particle** set you assigned to the Input is a multi-element (flexible and/or discrete-breakable) Custom Fiber, at least one Segment should be defined as frozen. (This shape is also known as a frozen Fiber (p. 1391); see also About Adding and Editing Particle Inputs (p. 708).) Important: All `release` times in the Particle Custom Inlet definition file must either be defined as 0 (zero) or the `release` field itself must be omitted from the definition file.

2. Ensure the Motion Frame you want to apply has been created. (See also Create a Motion Frame (p. 569).)

3. From the **Data** panel, under **Inputs**, select the Particle Custom Inlet to which you want to apply a Motion Frame.

4. From the **Data Editors** panel, select the **Particle Custom Inlets** tab, and then from the **Motion Frame** list, select the Motion Frame you want.

Tips:

- Even though you are able to use the Motion Preview window to see and test the movement in 3D, you will only be able to see the axes of the frame move and not the Custom Fibers. (See also Preview a Motion in 3D (p. 573).)

- To apply more than one movement to a single Particle Custom Inlet, add multiple motions to a single Motion Frame, or create a nested (child) Motion Frame and then apply the child frame to the Particle Custom Inlet. (See also Create a Motion Frame (p. 569).)

See Also:

- About Creating and Applying Motion Frames (p. 533)

CREATE AND MODIFY A MOTION PREVIEW WINDOW

A Motion Preview Window is where you preview your motions to see how they affect the geometries in your simulation. Previewing is typically done after you have assigned the motion frame to the geometry but before you process the simulation.

Like other types of Rocky windows, you are able to define how the objects within the window and the window itself appears and function on screen. After you set up your Motion Preview window, you can choose to reuse the zoom, rotation, and pan settings in another window by saving and applying a Custom Camera Preset. You can also use the Motion Preview window as a basis for creating an animation of your motions.

What do you want to do?

- Learn more About Motion Preview Windows (p. 572)

- Preview a Motion in 3D (p. 573)

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

- Change a Motion Preview Window (p. 575)
- Save a View with Custom Cameras (p. 1093)
- Create and Save an Animation (p. 1166)

See Also:

- About Creating and Applying Motion Frames (p. 533)

About Motion Preview Windows

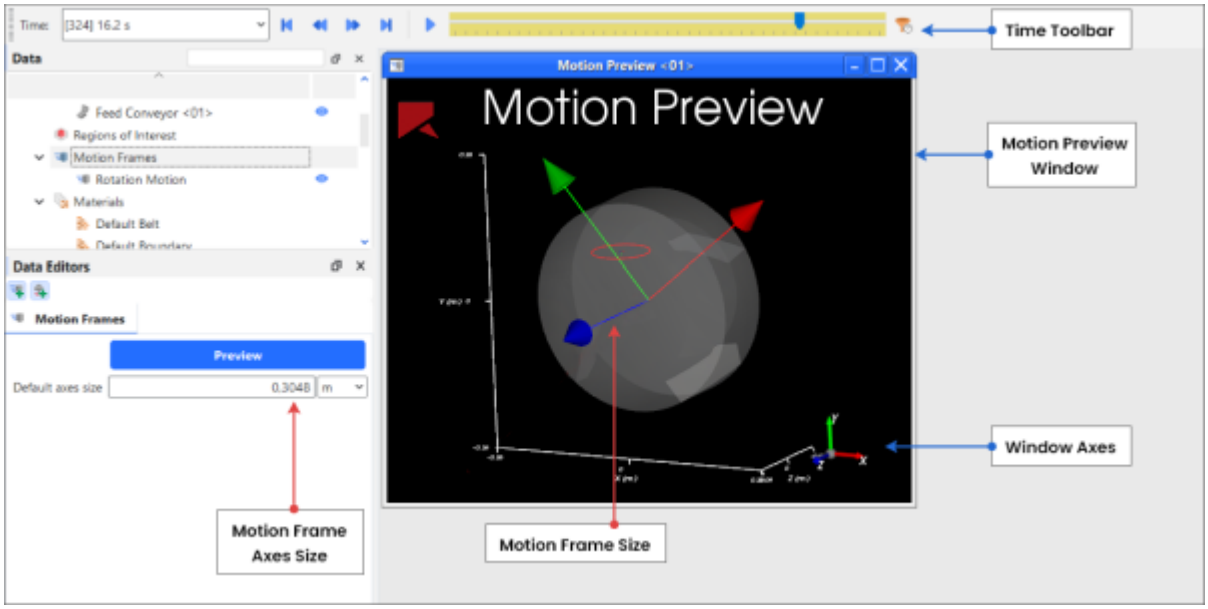
After you have defined one or more Motion Frames and have assigned them to your geometry components, a Motion Preview window is where you preview the geometry motions that you have defined. Previewing these motions before you process your simulation can allow you to catch and modify errors in your Motion Frame setup before particle calculations are introduced.

Notes:

- In this version of Rocky, the Motion Preview window will preview the effects of gravity and any additional (prescribed) forces/moments that you have defined for free body motions, but will not be able to predict motions as a result of interactions with particles.
- Even though you are able to preview the motion assigned to a Custom Input, only the axes for the frame will move; the particles themselves will not be visible in the preview.
- Every Motion Frame with a free body motion defined must first be associated with a geometry component in order to preview it without error on the Motion Preview window.

After you set up your Motion Frames and apply them to the components you want to move (see also About Creating and Applying Motion Frames (p. 533)), you use the yellow-highlighted Time Toolbar to "play" the preview (Figure 1). (See also the *Previewing Motions Using the Time Toolbar* section in the About the Time Toolbar (p. 1279) topic.)

Figure 3.90: Components of a Motion Preview Window



Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

The length of the preview allowed by the Time Toolbar is based upon the **Simulation Duration** value you set in the **Solver** | **Time** tab (see also About Solver Parameters (p. 464)) with an upper limit of no more than 30,000 time steps (p. 1395).

Within the Motion Preview window, each Motion Frame will be represented by its own set of axes. This axes set is tied to the local or parent coordinate of the frame and should not be confused with the set of window axes, which represents only the current orientation of the window itself. (See also About Using the Window Editors Panel to Change the Window Axes Displays (p. 1071).)

Tip:

You can change the size of the Motion Frame axes through the **Default Axes Size** parameter. This is located on the **Motion Frames** tab of the main **Motion Frames** entity. (See also the *Table 1: Motion Frames parameters (main entity)* section in the About Creating and Applying Motion Frames (p. 533) topic.)

As with other windows in Rocky, there are various ways you can change what appears in a Motion Preview window, including fonts, overlays, background colors, and grid lines. If you want to share your motions outside of Rocky, you can also use the Motion Preview window as a basis for creating animations.

After a simulation is processed, Motion Preview windows are still viewable but not as useful for post processing as a 3D View window. This is because after a simulation is processed, both a 3D View and a Motion Preview window will show geometry movements, but only the 3D View window will also show particles. (See also About 3D View Windows (p. 1040).)

What do you want to do?

- Preview a Motion in 3D (p. 573)
- Change a Motion Preview Window (p. 575)
- Learn more About Creating and Applying Motion Frames (p. 533)
- Create and Save an Animation (p. 1166)

Preview a Motion in 3D

Notes:

- In this version of Rocky, the Motion Preview window will preview the effects of gravity and any additional (prescribed) forces/moments that you define for free body motions, but will not be able to predict motions as a result of interactions with particles.
- Even though you are able to preview the motion assigned to a Custom Input, only the axes for the frame will move; the particles themselves will not be visible in the preview.
- Every Motion Frame with a free body motion defined must first be associated with a geometry component in order to preview it without error on the Motion Preview window.

1. Ensure that the motion you want to preview has been defined in a Motion Frame. (See also Create a Motion Frame (p. 569)).

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

2. Ensure that the geometry or Custom Input you want to move has been assigned the Motion Frame you want. (See also Apply a Motion Frame to an Imported Geometry (p. 569) or Apply a Motion Frame to a Custom Input (p. 571).) Tip:Although you can preview a motion without first assigning it to a component, for geometries especially, it helps to see the movement if the Motion Frame is assigned to a geometry before previewing.

3. From the **Data** panel, select **Motion Frames**, and then from the **Data Editors** panel, click **Preview**. A new Motion Preview window appears in the Workspace.

4. Use the buttons on the Time toolbar (see also About the Time Toolbar (p. 1279)) to play a preview of the motion as it might appear during the simulation.

Tip:

The time step slider turns yellow to indicate when it is being used to preview motion. If the time step slider is not yellow, it is being used to show simulated time steps as usual.

Note:

In preview mode, the time step slider is limited by the **Simulation Duration** value you set in the **Solver** | **Time** tab (see also About Solver Parameters (p. 464)) with an upper limit of no more than 30,000 time steps (p. 1395).

Tips:

- Zoom, pan, tilt, and fit the image just as you would in a 3D View window by using your mouse, keyboard, an/or the Camera Visualization toolbar. (See also Using the Mouse, Keyboard, or Toolbar to Change the View (p. 1043).)
- Show/hide the individual Motion Frames you want previewed by using eye icons on the **Data** panel. (See also Show/Hide Components by Using Eye icons and Checkboxes (p. 113).)
- Change attributes of the geometries and motion objects shown within the window by using the Coloring tab. (See also About Using the Coloring Tab to Change the Motion Preview for a Frame (p. 579).)
- Change the attributes of the window itself, such as background colors, by using the Windows Editors panel. (See also About Using the Window Editors Panel to Change the Selected Motion Preview Window (p. 575).)
- For motions involving high frequency vibrations, ensure that you have set the **Simulation** Output Frequency (on the **Solver** | **Time** tab) low enough to properly preview the motion. (See also About Solver Parameters (p. 464).)

See Also:

- Change a Motion Preview Window (p. 575)
- About Creating and Applying Motion Frames (p. 533)

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

Change a Motion Preview Window

There are several ways you can change what you see in the Motion Preview window once you have created it, as explained below.

- **Zoom, pan, tilt, and fit the image** by using your mouse, keyboard, and/or the Camera Visualization toolbar.
- **Change the colors and data attributes of the image**, including the faces, edges, nodes, or vectors being displayed by using the Coloring tab on the Data Editors panel.
- **Show/hide the simulation entity you want displayed** by using eye icons on the Data panel.
- **Change the attributes of the window itself**, such as background colors, grid lines, and legend and caption details; and display options for the window axes by using the Window Editors panel.

You can also remove, rename, and show/hide a Motion Preview window just like any other window in Rocky.

What would you like to do?

- Learn more about Using the Mouse, Keyboard, or Toolbar to Change the View (p. 1043)
- Learn more about Using the Coloring Tab to Change a Particles Details Window (p. 694)
- Learn more about Using the Window Editors Panel to Change a Motion Preview Window (p. 682)
- Show/Hide Components by Using Eye icons and Checkboxes (p. 113)

See Also:

- Save a View with Custom Cameras (p. 1096)

USING THE WINDOW EDITORS PANEL TO CHANGE A MOTION PREVIEW WINDOW

From the Window Editors panel, you can modify just the selected Motion Preview window to the change attributes including the background color, font color/sizes, and synchronize the time steps displayed when two or more view windows are open. You may also use the Window Editors panel to adjust the axes display settings for all the windows in your project.

What would you like to do?

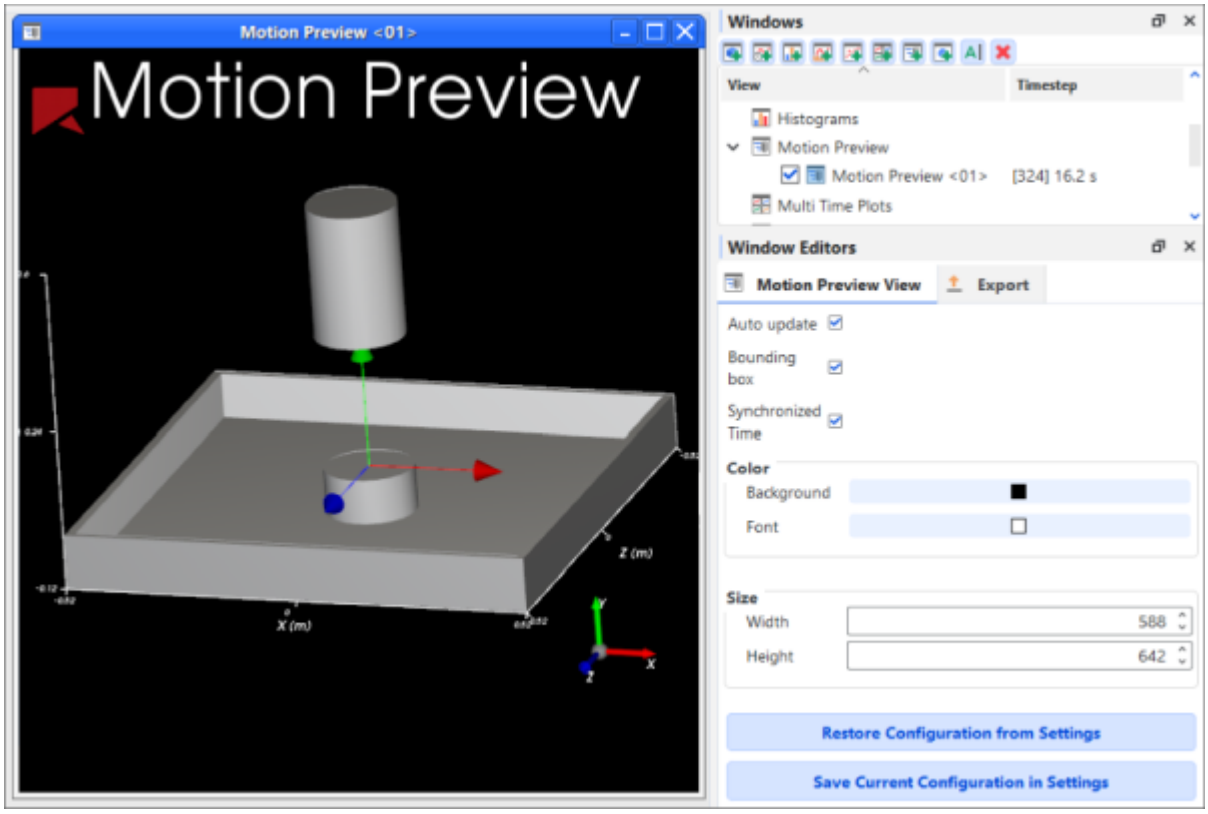
- Learn more About Using the Window Editors Panel to Change the Selected Motion Preview Window (p. 575)
- Change Window Display and Data Update Options for the Selected Motion Preview Window (p. 578)
- Learn more About Using the Window Editors Panel to Change the Window Axes Displays (p. 1071)
- Change the Axes Display for all Rocky Windows (p. 1074)
- Using the Coloring Tab to Change a Motion Preview Window (p. 579)

ABOUT USING THE WINDOW EDITORS PANEL TO CHANGE THE SELECTED MOTION PREVIEW WINDOW

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

When you use the Window Editors panel to change the selected Motion Preview window, you have the ability to change how items in the window appear on the screen. These items include background color, and font color/sizes (Figure 1). You can also change how multiple windows of the same type respond to changes in time steps.

Figure 3.91: Motion Preview View tab options on the Window Editors panel



Tips:

- You can also use Rocky Preferences to change the background color and fonts for all the Motion Preview windows in your project, instead of just the selected one. (See the "Preferences for Windows Properties" section in the About Setting Global Preferences (p. 227) topic.)
- You can also change the window axes display for all windows in your project, including Motion Preview windows. (See also About Using the Window Editors Panel to Change the Window Axes Displays (p. 1071).) Note: The window axes is different from the motion frame axes, which has its own display settings. (See also About Creating and Applying Motion Frames (p. 533).)

In this section, we will cover only the items on the **Motion Preview View** tab. Note: The **Export** tab will be covered in the About Exporting Data and Images Out of Rocky (p. 1155) topic.

You use the **Motion Preview View** tab on the **Window Editors** panel to change how the window appears and how the data displayed within it is updated. These options include changing display items like the background color; labels, lines, and axes colors; color scale label sizes; and window height and width. In addition, you can choose to show or hide the **Bounding box**, which displays measurements illustrating the simulation boundaries; use **Auto update** to enable or prevent the view from being updated when data affecting it is changed; or choose to have multiple views updated with the same time step by using **Synchronized Time**.

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

MOTION PREVIEW VIEW TAB SETTING DEFINITIONS

See the image above and table below to understand more about how you can use the Window Editors panel to change a Motion Preview window.

Table 1: Motion Preview View tab options on the Window Editors panel

Setting	Description	Range
Auto update	When enabled, any change made to the display settings of the selected Motion Preview window or to the items affecting the data or settings calculated and displayed within the Motion Preview, including the time step or Geometries displayed, will be updated in the Motion Preview window automatically. Because automatically updating the window with each separate change can be computationally intensive, you can clear this option to have all calculated items and more computationally intensive display options remain unchanged in the window until you enable **Auto update** again. **Tip:** You will know when the calculations in a Motion Preview window are not being updated when you see a thick red border around the window.	Turns on or off
Bounding box	When enabled, displays measurements illustrating the simulation boundaries in the selected Motion Preview window.	Turns on or off
Synchronized Time	When enabled and the window (or any other window with this checkbox enabled) is selected, the details shown in this (and any other Synchronized Time) window will be updated when the current time step is changed on the Time toolbar. (See also About the Time Toolbar.) When cleared and the window is selected, only this window will be updated when the current time step is changed on the Time toolbar. **Tip:** To keep the time step synchronized between multiple **Motion Preview** or other windows, ensure each window has this option enabled.	Turns on or off
Color Background	Enables you to change the color that appears behind the geometries in the Motion Preview window.	Options limited by the choices in the **Select Color** dialog

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

Setting	Description	Range
Color Font	Enables you to change the color of the labels, borders, and axes lines displayed in the Motion Preview window.	Options limited by the choices in the **Select Color** dialog
Size Width	Enables you to numerically change how wide the selected window is. Useful in cases where you need exact dimensions for image consistency, or when generating animations.	Whole positive values between 123 and 5000.
Size Height	Enables you to numerically change how tall the window is. Useful in cases where you need exact dimensions for image consistency, or when generating animations.	Whole positive values between 0 (zero) and 5000.
Restore Configuration from Settings	Clicking this button replaces the values set on the tab with the ones that have been saved to the (internal) Rocky Settings folder. (See also Rocky File Types and Folders.)	(Button selection)
Save Current Configuration in Settings	Clicking this button overwrites the values that have been saved to the (internal) Rocky Settings folder with the ones currently set on the tab. (See also Rocky File Types and Folders.)	(Button selection)

What would you like to do?

- Change Window Display and Data Update Options for the Selected Motion Preview Window (p. 578)
- Learn more About Exporting Data and Images Out of Rocky (p. 1155)

See Also:

- Preview a Motion in 3D (p. 573)
- About Creating and Applying Motion Frames (p. 533)
- About Using the Window Editors Panel to Change the Window Axes Displays (p. 1071)
- Change a Motion Preview Window (p. 575)

CHANGE WINDOW DISPLAY AND DATA UPDATE OPTIONS FOR THE SELECTED MOTION PREVIEW WINDOW

1. Ensure the **Window Editors** panel is visible. (From the **View** menu, click **Window Editors**.)
2. From the Workspace (p. 1396), select the Motion Preview window that you want to change. (See also Preview a Motion in 3D (p. 573).)
3. From the **Window Editors** panel, select the **Motion Preview View** tab and then change the options you want.

Tips:

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

- You can also open the **Window Editors** panel for a window by right-clicking an empty space within the Motion Preview window (for example, the background behind the geometries), and then clicking **Settings**.
- You can also use Rocky Preferences to change the background color and fonts for all the Motion Preview windows in your project, not just selected one. (See the "Preferences for Windows Properties" section in the About Setting Global Preferences (p. 227) topic.)
- To reuse in future projects the settings you made to this tab, click the **Save Current Configuration in Settings** button.
- To apply settings you have already saved (either by previously using the **Save Current Configuration in Settings** button or by saving selections within the **Preferences** dialog) to this tab, click the **Restore Configuration from Settings** button.

See Also:

- About Using the Window Editors Panel to Change the Selected Motion Preview Window (p. 575)
- About Creating and Applying Motion Frames (p. 533)
- About Using the Window Editors Panel to Change the Window Axes Displays (p. 1071)
- Change a Motion Preview Window (p. 575)

USING THE COLORING TAB TO CHANGE A MOTION PREVIEW WINDOW

The Coloring tab, which is located on the Data Editors panel for all simulation entities and User Processes, enables you to change the image of the items being displayed within your Motion Preview window. These include the geometry components and any user processes created from those geometry components. It also includes individual Motion Frames whose Coloring tab shows options related to the axes of the Frame that is selected in the Data panel.

What would you like to do?

- Learn more About Using the Coloring Tab to Change the Motion Preview for a Frame (p. 579)
- Learn more About Color Scales (p. 1086)
- Use the Coloring Tab to Change the Motion Preview for a Frame (p. 582)

See Also:

- Change a Motion Preview Window (p. 575)

ABOUT USING THE COLORING TAB TO CHANGE THE MOTION PREVIEW FOR A FRAME

Using the options on the Coloring tab is one way you can change the appearance of the items being displayed in a Motion Preview window. Visualization options for the Faces, Edges, and Nodes can be modified with the Coloring tab for any of the following entities:

- Default Conveyors or Imported Geometry Components (p. 480)
- Inlets

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

- Any User Process (p. 1174) created from any of the above entities

In addition, when an individual Motion Frame is selected in the Data panel, the Coloring tab for that entity shows options related to the axes of the selected Frame (Figure 1).

PARAMETER DEFINITIONS

See the images and tables below to help you understand how to use the Coloring tab to change a view for selected entity.

Figure 3.92: Coloring tab when a Motion Frame is selected in the Data panel AND a Motion Preview window is selected

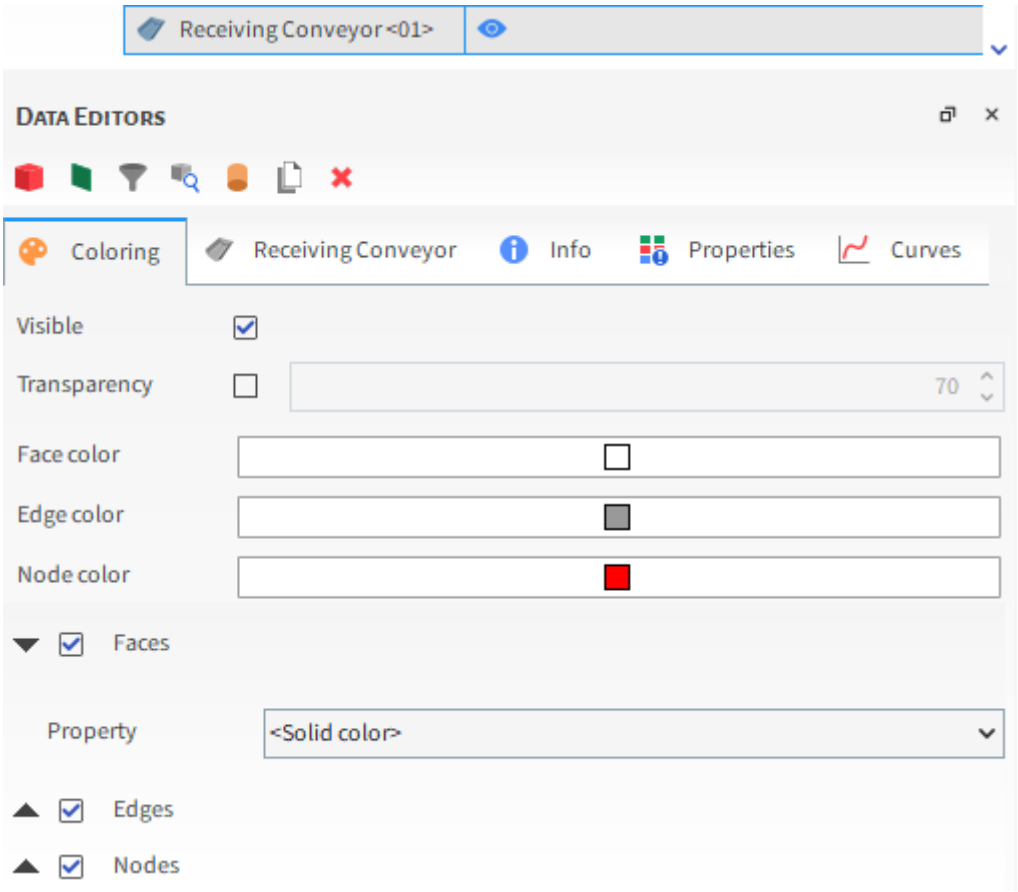


Table 1: Coloring tab parameter options for a Motion Frame when a Motion Preview window is selected

Setting	Description	Range
Visible	When enabled, shows the selected entity in the active view window. **Note:** This is the same as using the Data panel eye icon. (See also Show/Hide Components by Using Eye icons and Checkboxes (p. 113).)	Turns on or off

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

Setting	Description	Range
Inherit Axes Size?	When enabled, allows the selected Frame to use the same axes size as what was set in the main Motion Frames entity (see also About Creating and Applying Motion Frames to Imported Geometries (p. 533)). When cleared, enables you to set a unique axes size for the selected Motion Frame.	Turns on or off
Axes Size	When the **Inherit Axes Size?** checkbox is cleared, this defines the custom axes size for the selected Frame.	Positive value

Figure 3.93: Example Coloring tab in the Data Editors panel showing a geometry component

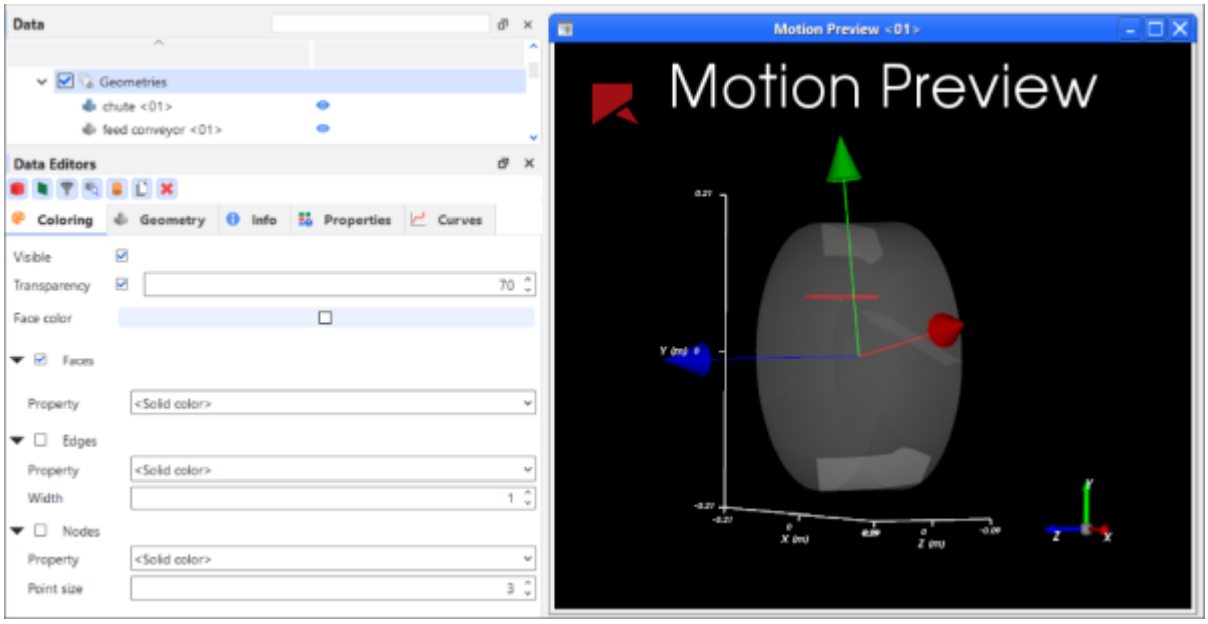


Table 2: Coloring tab parameter options for a geometry component when a Motion Preview window is selected

Setting	Description	Range
Visible	When enabled, shows the selected entity in the active view window. **Note:** This is the same as using the Data panel eye icon. (See also Show/Hide Components by Using Eye icons and Checkboxes (p. 113).)	Turns on or off
Transparency	Enables boundaries to be drawn transparently in the selected view window. A value of 0 (zero) makes the faces completely solid or opaque. A value of 100 makes the faces completely transparent or hidden.	Turns on or off; 0-100
Color	When **Faces**, **Edges**, and/or **Nodes** are selected and the **Property** is <Solid Color>, this enables the selected display type to be colored in the one solid color chosen. Note: For	Options limited by the **Select Color** dialog

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

Setting	Description	Range
	Faces, **Transparency** options affect how opaque this color appears in the view.	
Faces	Enables the entire surface of the entity to be visible and affected by the various Faces coloring options.	Turns on or off
Property	Three separate lists under **Faces**, **Edges**, and **Nodes** respectively provide properties and other color options that apply to the display type within which the list is contained. Specifically: • **<Solid Color>** applies a single color to the display type. • The remaining properties come directly from the component's Properties tab. Selecting one of these properties colors display type in a range according to the data collected for the current time step. (See also About Properties (p. 933).)	<Solid color>; List of properties automatically generated from the Properties tab
Show on Node?	When **Faces** is enabled and a property is selected from the **Property** list under **Faces**, this enables the visualization of the selected property to be evenly interpolated across the entity for a smoother visualization.	Turns on or off
Edges	Enables only the edges bordering the components making up the entity boundaries visible and affected by the various Edges coloring options.	Turns on or off
Width	When **Edges** is selected, this changes the thickness of the lines used to draw the borders. Entering 1 will provide the narrowest line.	Positive values
Nodes	Enables the points marking the intersections of the entity's individual components to be visualized as dots and affected by the various Nodes coloring options.	Turns on or off
Point Size	When **Nodes** is selected, this changes the size of the dots used to draw the nodes. Entering 1 will provide the smallest dot.	Positive values

What would you like to do?

- Use the Coloring Tab to Change the Motion Preview for a Frame (p. 582)

See Also:

- About Creating and Applying Motion Frames (p. 533)
- Change a Motion Preview Window (p. 575)

USE THE COLORING TAB TO CHANGE THE MOTION PREVIEW FOR A FRAME

1. From the Workspace (p. 1396), select the Motion Preview window to which you want to apply changes. (See also Preview a Motion in 3D (p. 573).)

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

2. From the **Data** panel, do one of the following:
   - Under **Geometries**, select the geometry component or inlet that you want to change.
   - Under **Motion Frame**, select the Frame you want to change.
3. From the **Data Editors** panel, select the **Coloring** tab, and then change the settings that you want. The changes you make are shown automatically in the Motion Preview window.

See Also:

- About Using the Coloring Tab to Change the Motion Preview for a Frame (p. 579)
- Change a Motion Preview Window (p. 575)
- About Motion Preview Windows (p. 572)

USE MOTION FRAMES TO CREATE SPECIFIC MOVEMENTS

Once you know how to create and apply Motion Frames to an imported geometry (see also About Creating and Applying Motion Frames to Imported Geometries (p. 533)), there are many different types of movements you can create. Use this section to learn more about how to create some specific movements for your geometries.

What would you like to do?

- Use Motion Frames to Enable a Geometry to Move Linearly (p. 583)
- Use Motion Frames to Enable a Geometry to Rotate Around its Center Point (e.g. a Rotating Mill Slice) (p. 584)
- Use Motion Frames to Enable a Regularly Swinging Boundary (e.g. a Pendulum) (p. 585)
- Use Motion Frames to Enable a Vibrating Boundary (p. 585)
- Use Motion Frames to Enable a Geometry to Freely Move Linearly with an Added Force (p. 587)
- Use Motion Frames to Enable a Geometry to Freely Rotate Around its Center Point (p. 587)
- Use Motion Frames to Enable a Geometry to Freely Rotate Around a Non-Center Point with Added Torque (e.g., a Gate or Rubber Curtain) (p. 588)
- Use a Cone Crusher Frame to Enable a Geometry to Adopt a Crushing Motion (p. 589)
- Use Motion Frames to Enable Multiple Movements of a Geometry in one Simulation (p. 589)
- Use Motion Frames to Enable Geometry Motions to be Repeated (p. 590)

Use Motion Frames to Enable a Geometry to Move Linearly

1. Ensure the geometry you want to move has already been imported.
2. Create a Motion Frame as you normally would (see also Create a Motion Frame (p. 569)), but under **Motions** (step 3b), ensure that you do all of the following:

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

a. Determine when you want the motion to start and stop by entering the values you want for **Start Time** and **Stop Time**.

b. From the **Type** list, select **Translation**.

c. From the **Input** list, do one of the following:

- To set only a fixed velocity with no acceleration, select **Fixed Velocity**, and then enter the **Velocity** you want in the X, Y, and Z axes respectively.
- To set both the initial and final velocity and have Rocky compute the acceleration required to achieve those values, select **Initial and Final Velocity** and then enter the **Initial Velocity** and **Final Velocity** you want in the X, Y, and Z axes respectively.
- To set both the initial velocity and the acceleration values and then have Rocky compute the final velocity that results from those values, select **Initial Velocity and Acceleration** and then enter the **Initial Velocity** and **Acceleration** values you want in the X, Y, and Z axes respectively.

3. Apply the new Motion Frame to the geometry you want to move. (See also Apply a Motion Frame to an Imported Geometry (p. 569).)

4. Use the Motion Preview window to see and test the movement in 3D. (See also Preview a Motion in 3D (p. 573).)

See Also:

- About Creating and Applying Motion Frames (p. 533)

Use Motion Frames to Enable a Geometry to Rotate Around its Center Point

1. Ensure the geometry you want to move has already been imported.

2. Create a Motion Frame as you normally would (see also Create a Motion Frame (p. 569)), but ensure that you do all of the following:

a. For **Relative Position**, enter the location of the geometry's center point.

b. Under **Motions** (step 3b), do all of the following:

i. Determine when you want the motion to start and stop by entering the values you want for **Start Time** and **Stop Time**.

ii. From the **Type** list, select **Rotation**.

iii. Enter the **Initial Angular Velocity** you want in the X, Y, and Z axes respectively.

iv. If you want velocity affected by acceleration, enter the **Angular Acceleration** you want in the X, Y, and Z axes respectively.

3. Apply the new Motion Frame to the geometry you want to move. (See also Apply a Motion Frame to an Imported Geometry (p. 569).)

4. Use the Motion Preview window to see and test the movement in 3D. (See also Preview a Motion in 3D (p. 573).)

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

See Also:

- About Creating and Applying Motion Frames (p. 533)

Use Motion Frames to Enable a Regularly Swinging Boundary (e.g. a Pendulum)

Note:

Use this procedure when the swinging motion is applied to the geometry in a predictable and regular manner. To enable a boundary that swings freely based upon particles acting upon it, see the procedure Use Motion Frames to Enable a Geometry to Rotate Freely Around a Non-Center Point (e.g., a Gate or Rubber Curtain) (p. 588).

---

1. Ensure the geometry you want to move has already been imported.
2. Create a Motion Frame as you normally would (see also Create a Motion Frame (p. 569)), but under **Motions** (step 3b), ensure that you do all of the following:
   a. Determine when you want the motion to start and stop by entering the values you want for **Start Time** and **Stop Time**.
   b. From the **Type** list, select **Periodic Rotation (Pendulum)**.
   c. Enter the **Initial Frequency** and **Initial Amplitude**.
   d. For **Direction**, enter the X, Y, Z vector components defining the 2D plane upon which the swinging will occur.
   e. For **Initial Phase**, enter the starting angular degree (Plane Angle) you want to use.
   f. If you want the frequency or amplitude to vary per unit of time for the selected motion, enter what you want for **Frequency Variation** and **Amplitude Variation**.
3. Apply the new Motion Frame to the geometry you want to move. (See also Apply a Motion Frame to an Imported Geometry (p. 569).)
4. Use the Motion Preview window to see and test the movement in 3D. (See also Preview a Motion in 3D (p. 573).)

See Also:

- Use Motion Frames to Create Specific Movements (p. 583)
- About Creating and Applying Motion Frames (p. 533)

Use Motion Frames to Enable a Vibrating Boundary

1. Ensure the geometry you want to move has already been imported.
2. Create a Motion Frame as you normally would (see also Create a Motion Frame (p. 569)), but under **Motions** (step 3b), ensure that you do all of the following:
   a. Determine when you want the motion to start and stop by entering the values you want for **Start Time** and **Stop Time**.

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

b. From the **Type** list, select **Periodic Translation (Vibration)**.

c. Enter the **Initial Frequency** and **Initial Amplitude**.

d. For **Direction**, enter the X, Y, Z vector components defining the direction over which the vibrating motion will occur.

e. For **Initial Phase**, enter the starting angular degree (Plane Angle) you want to use.

f. If you want the frequency or amplitude to vary per unit of time for the selected motion, enter what you want for **Frequency Variation** and **Amplitude Variation**.

3. Apply the new Motion Frame to the geometry you want to move. (See also Apply a Motion Frame to an Imported Geometry (p. 569).)

4. Use the Motion Preview window to see and test the movement in 3D. (See also Preview a Motion in 3D (p. 573).) Tip:For motions involving high frequency vibrations, ensure that you have set your **Solver** | **Output Frequency** low enough to properly preview the motion. (See also About Solver Parameters (p. 464).)

See Also:

- Use Motion Frames to Create Specific Movements (p. 583)
- About Creating and Applying Motion Frames (p. 533)
- Workshop: 03 - Vibrating Screen

Use Motion Frames to Enable a Geometry to Freely Move Linearly

1. Ensure the geometry you want to move has already been imported.

2. Create a Motion Frame as you normally would (see also Create a Motion Frame (p. 569)), but under **Motions** (step 3b), ensure that you do all of the following:

   a. Determine when you want the motion to start and stop by entering the values you want for **Start Time** and **Stop Time**.

   b. From the **Type** list, select **Free Body Translation**.

   c. From the **Free Motion Direction** list, select the axis or axes that you want free motion enabled. Tip:To have Free Motion enabled in every possible direction, select **All directions**.

3. Apply the new Motion Frame to the geometry you want to move. (See also Apply a Motion Frame to an Imported Geometry (p. 569).) Note: With free body motions, missing this step will result in errors when you later try to either preview the motion or process the simulation.

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

4. Use the Motion Preview window to see and test the effects of gravity. (See also Preview a Motion in 3D (p. 573).)

Note:

Because the movement of a geometry with Free Motion is directly affected by particle interactions yet to be calculated, the Motion Preview window will only be able to preview the effects of gravity and any additional (prescribed) forces/moments you have defined.

See Also:

- Use Motion Frames to Create Specific Movements (p. 583)
- About Creating and Applying Motion Frames (p. 533)

Use Motion Frames to Enable a Geometry to Freely Move Linearly with an Added Force

1. Ensure the geometry you want to move has already been imported.
2. Create a Motion Frame with a **Free Body Translation** Motion **Type**.
3. Add a second Motion to the selected Frame by clicking the **Add Motion** button, and then do all of the following:
   a. Determine when you want the motion to start and stop by entering the values you want for **Start Time** and **Stop Time**.
   b. From the **Type** list, select **Additional Force**.
   c. For **Force Value**, enter the X, Y, and Z components defining the amount of additional, applied force you want acted upon the selected Frame.

#. Apply the new Motion Frame to the geometry you want to move. (See also Apply a Motion Frame to an Imported Geometry (p. 569).) Note: With free body motions, missing this step will result in errors when you later try to either preview the motion or process the simulation. #. Use the Motion Preview window to see and test the effects of gravity and the additional (prescribed) forces/moments that you defined. (See also Preview a Motion in 3D (p. 573).)

Note:

Because the movement of a geometry with Free Motion is directly affected by particle interactions yet to be calculated, the Motion Preview window will only be able to preview the effects of gravity and any additional (prescribed) forces/moments that you have defined.

See Also:

- Use Motion Frames to Create Specific Movements (p. 583)
- About Creating and Applying Motion Frames (p. 533)

Use Motion Frames to Enable a Geometry to Freely Rotate Around its Center Point

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

1. Ensure the geometry you want to move has already been imported.
2. Create a Motion Frame as you normally would (see also Create a Motion Frame (p. 569)), but ensure that you do all of the following:
   a. For **Relative Position**, enter the location of the geometry's center point.
   b. Under **Motions** (step 3b), do all of the following:
      i. Determine when you want the motion to start and stop by entering the values you want for **Start Time** and **Stop Time**.
      ii. From the **Type** list, select **Free Body Rotation**.
      iii. From the **Free Motion Direction** list, select the axis or axes that you want free motion enabled. Tip:To have Free Motion enabled in every possible direction, select **All directions**.
3. Apply the new Motion Frame to the geometry you want to move. (See also Apply a Motion Frame to an Imported Geometry (p. 569).) Note: With free body motions, missing this step will result in errors when you later try to either preview the motion or process the simulation.
4. Use the Motion Preview window to see and test the effects of gravity. (See also Preview a Motion in 3D (p. 573).)

Note:

Because the movement of a geometry with Free Motion is directly affected by particle interactions yet to be calculated, the Motion Preview window will only be able to preview the effects of gravity and any additional (prescribed) forces/moments you have defined.

See Also:

- Use Motion Frames to Create Specific Movements (p. 583)
- About Creating and Applying Motion Frames (p. 533)

Use Motion Frames to Enable a Geometry to Freely Rotate Around a Non-Center Point with Added Torque (e.g., a Gate or Rubber Curtain)

1. Ensure the geometry you want to move has already been imported.
2. Create a Motion Frame with a **Free Body Rotation** Motion **Type** (see also Use Motion Frames to Enable a Geometry to Freely Rotate Around its Center Point (p. 587)) but be sure to set the Frame **Relative Position** to be the coordinate location of the point you want the geometry to rotate around.
3. Add a second Motion to the selected Frame by clicking the **Add Motion** button, and then do all of the following:
   a. Determine when you want the motion to start and stop by entering the values you want for **Start Time** and **Stop Time**.
   b. From the **Type** list, select **Additional Moment**.

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

c. For **Moment Value**, enter the X, Y, and Z components defining the amount of additional, applied moment (torque) you want acted upon the selected Frame.

4. Apply the new Motion Frame to the geometry you want to move. (See also Apply a Motion Frame to an Imported Geometry (p. 569).) Note: With free body motions, missing this step will result in errors when you later try to either preview the motion or process the simulation.

5. Use the Motion Preview window to see and test the effects of gravity and the additional (prescribed) forces/moments that you defined. (See also Preview a Motion in 3D (p. 573).)

Note:

Because the movement of a geometry with Free Motion is directly affected by particle interactions yet to be calculated, the Motion Preview window will only be able to preview the effects of gravity and any additional (prescribed) forces/moments that you have defined.

See Also:

- Use Motion Frames to Create Specific Movements (p. 583)
- About Creating and Applying Motion Frames (p. 533)

Use a Cone Crusher Frame to Enable a Geometry to Adopt a Crushing Motion

1. Ensure that both the mantle and shaft geometries you want to move have already been imported.

2. Create a Cone Crusher Frame as you normally would (see also Create a Cone Crusher Frame (p. 567)).

3. Apply the new Cone Crusher Frame to the two geometries you want to move. (See also Apply a Cone Crusher Frame to Custom Mantle and Shaft Geometries (p. 568).)

Note:

Due to the free body motions involved, you cannot currently use the Motion Preview window to see and test the Cone Crusher Frames in 3D. (See also Preview a Motion in 3D (p. 573).)

See Also:

- About Creating and Applying Cone Crusher Frames to Imported Geometries (p. 565)

Use Motion Frames to Enable Multiple Movements of a Geometry in One Simulation

1. Ensure the geometry you want to move has already been imported.

2. Create a Motion Frame as you normally would (see also Create a Motion Frame (p. 569)).

3. Do one of the following:

   - To create an additional motion within the selected Motion Frame, click the **Add Motion** button, and set up the additional motion as you normally would.

   - To create a separate Frame linked to the selected Motion Frame, do the following:

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

a. Click the **Add Motion Frame** button. A nested (child) Frame appears beneath the original Frame in the **Data** panel.

b. From the **Data** panel, select the new Frame, and then set up the motion as you normally would.

4. Apply the Motion Frame to the geometry you want to move. (See also Apply a Motion Frame to an Imported Geometry (p. 569).) Specifically:

   - For multiple motions within one Motion Frame, apply the single Motion Frame to the geometry. All motions within the Motion Frame will be applied.
   - For multiple nested Motion Frames, apply the lowest-level child Motion Frame to the geometry. All Motion Frames linked to the child, including their defined motions, will be applied.

5. Use the Motion Preview window to see and test the movement in 3D. (See also Preview a Motion in 3D (p. 573).)

See Also:

- Use Motion Frames to Create Specific Movements (p. 583)
- About Creating and Applying Motion Frames (p. 533)

Use Motion Frames to Enable Geometry Motions to be Repeated

1. Ensure the geometry you want to have periodic motions has already been imported.
2. Create a Motion Frame as you normally would (see also Create a Motion Frame (p. 569)), but but for part 3a, do the following:

   - Be sure that you enable the **Enable Periodic Motion** checkbox.
   - In the **Periodic Start Time** box, enter the amount of delay after the motion(s) **Start Time** and **Stop Time** to begin and end the motion to be repeated.
   - In the **Periodic Stop Time** box, enter the exact time you want the motion(s) to stop, regardless of what is set for its **Stop Time**.

     Tip:

     To ensure that the periodic motion continues for the entire simulation, keep this value set to the default (1000 s) or set it higher than your simulation duration.

   - In the **Period** box, enter how much of the original motion(s) will be repeated.

3. Continue setting up your Motion Frame as you normally would.
4. Use the Motion Preview window to see and test the movement in 3D. (See also Preview a Motion in 3D (p. 573).)

Tips:

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

- To have the periodic motion appear at simulation onset, keep both the **Start Time** and **Periodic Start Time** zero (0).

See Also:

- Use Motion Frames to Create Specific Movements (p. 583)
- About Creating and Applying Motion Frames (p. 533)

3.2.11. Create and Modify Material Compositions

Rocky enables you to specify unique properties for both solid and fluid materials. Conveyor belts, geometries, and particle sets and fluid elements used in your simulation can each have specific materials, allowing you to ensure that the various components being simulated interact with each other in as realistic a way as possible.

What would you like to do?

- Learn more About Modifying Solid Material Compositions (p. 591)
- Learn more About Modifying Fluid Material Compositions (p. 595)

See Also:

- Set Simulation Parameters (p. 414)

ABOUT MODIFYING SOLID MATERIAL COMPOSITIONS

By default, Rocky defines three Solid Material sets for each simulation (for Default Belt, Default Boundary and Default Particle). While you can modify the settings for these default Materials and also add additional Material definitions if you choose, you cannot remove them from the simulation.

Materials you define here will be used when defining the parameters for Geometries, Particles, and Materials Interactions.

Materials and External Modules

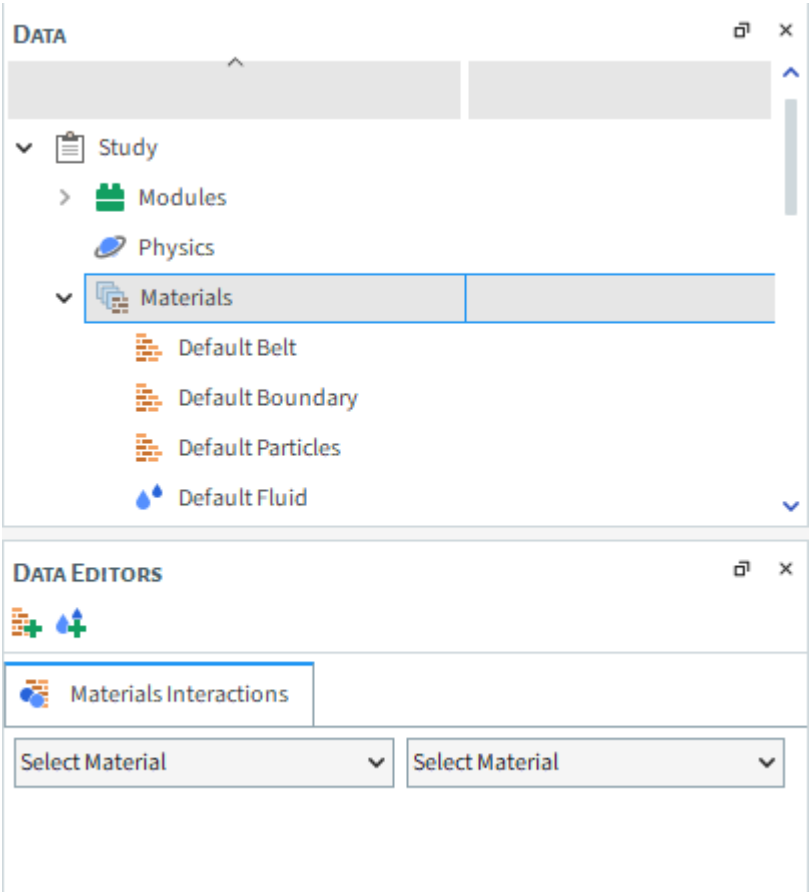
If you have enabled one or more external Modules (p. 1391) that affect your Materials settings, you might also have separate, module-specific settings that you can define. Refer to the Module's documentation (if provided) for more information. (See also Rocky Simulation Entities that can be Affected by Modules (p. 223).)

Materials Parameter Definitions

Use the figures and table below to help you understand the various parameters you can modify for Materials.

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

Figure 3.94: Materials parameters in the Data Editors panel



Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

Figure 3.95: Default Material parameters in the Data Editors panel

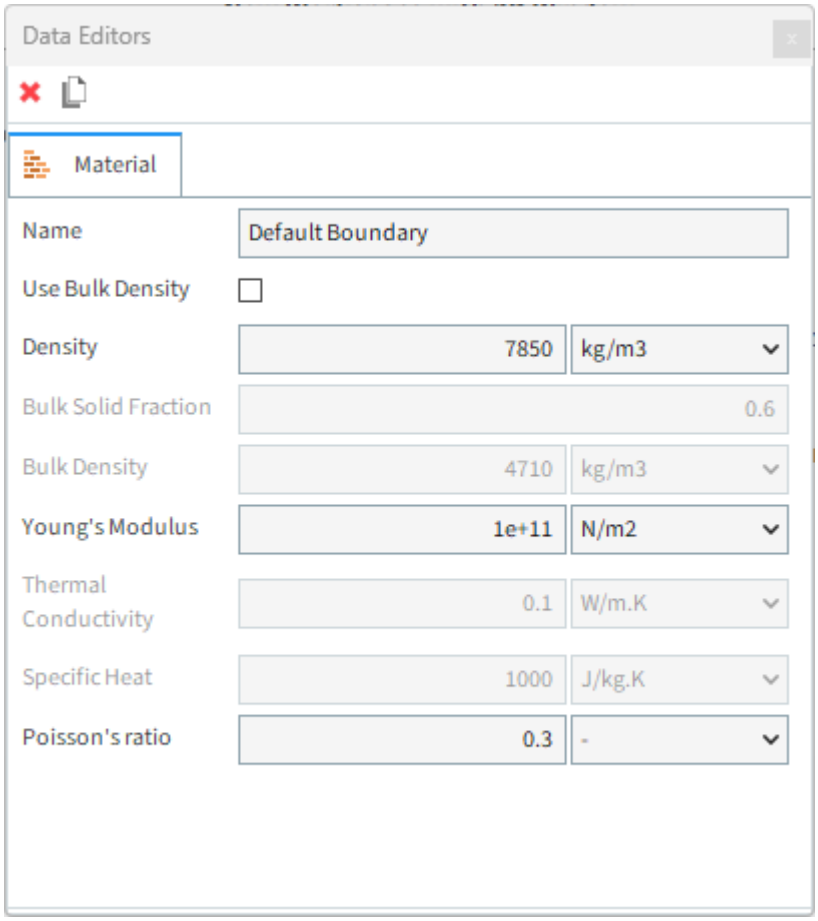


Table 1: Material parameter options (all available)

Setting	Description	Range
Name	Enables you to specify a unique identifier for the Material.	99 character limit
Use Bulk Density	When selected, the **Density** of the particles will be calculated using both the **Bulk Density** and the **Bulk Solids Fraction for Materials** values. When cleared, the **Density** value you enter directly will be used instead. **Note:** No matter what you choose here, only the **Density** value-either entered directly by you or calculated by Rocky (and shown grayed-out) based upon the bulk density values you provided-is used for DEM calculations.	Turns on or off
Density	The particle density or true density of a particulate solid or powder is the density of the material that makes up the particles. No	Positive values

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

Setting	Description	Range
	matter whether this value is entered by you or is calculated by Rocky (and shown grayed- out) based upon the bulk density values you provided, only this value will be used for DEM calculations.	
Bulk Solid Fraction	For all Materials that have the **Use Bulk Density** checkbox enabled, this number is applied to the (disabled) **Density** value of the Material. For all Materials that have the **Use Bulk Density** checkbox cleared, this number is applied to the (disabled) **Bulk Density** of the value. **Note:** One minus this value is the void fraction (also known as void ratio) of the Materials. (The default void fraction is 0.4.) Void fraction or porosity is a measure of the void spaces in a material. It is a fraction of the volume of voids over the total volume, between 0 and 1, or as a percentage between 0 and 100%.	Between 0 and 1
Bulk Density	The bulk density or apparent density of a material is defined as the mass of particles divided by the total volume they occupy. The total volume includes particle volume, inter-particle void volume, and internal pore volume. Therefore, bulk density is *not* an intrinsic property of a material; it can change depending upon how the material is handled and compacted; i.e., it depends upon the **Bulk Solid Fraction**. For example, a powder poured into a cylinder will have a particular bulk density; if the powder is then compacted by the application of a certain force over the particles, particles will move and stay closer together, resulting in a higher bulk density. Bulk density of granular materials is usually reported both as "freely settled" (or "poured" density) and "tapped" density, which is the bulk density of the material after a specified compaction process.	Positive values
Young's Modulus	A measure of the capacity of a material to withstand changes in length when under lengthwise tension or compression. Defines the relationship between force per unit area and proportional deformation, and is assigned as separate values to each particle and boundary material defined. Applied in contact stiffness calculations used in the models for normal and tangential forces for each pair of particle-to-particle or particle-to-boundary collisions. In general, the higher the value, the more accurate the results but the longer the processing time.	Positive values

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

Setting	Description	Range
Thermal Conductivity	When **Thermal Model** is enabled (see also About Physics Parameters (p. 417)), this sets the thermal conductivity value for the material.	Any value
Specific Heat	When **Thermal Model** is enabled (see also About Physics Parameters (p. 417)), this sets the specific heat value for the material.	Any value
Poisson's Ratio	This sets the Poisson's Ratio value for the material. **Note:** Even though you are always able to set this parameter, it will only be used in the following scenarios: • When the **Thermal Model** is enabled (see also About Physics Parameters (p. 417)) • When the **Hertzian Spring Dashpot** Normal Force is selected (see also About Physics Parameters (p. 417)) • When a **Fiber** Particle Shape is selected (see also About Adding and Editing Particle Sets (p. 602))	Any value **Tip:** Most materials will have a Poisson's Ratio value ranging between 0.0 and 0.5

See Also:

- Set Simulation Parameters (p. 414)
- Enable Thermal Modeling Calculations (p. 400)

ABOUT MODIFYING FLUID MATERIAL COMPOSITIONS

By default, Rocky defines a Fluid Material. While you can modify the settings for this default Material and also add additional Material definitions if you choose, you cannot remove it from the simulation.

Materials you define here will be used when defining the parameters for the Fluid phase, and its interactions.

Fluid Materials and External Modules

If you have enabled one or more external Modules (p. 1391) that affect your Materials settings, you might also have separate, module-specific settings that you can define. Refer to the Module's documentation (if provided) for more information. (See also Rocky Simulation Entities that can be Affected by Modules (p. 223).)

Fluid Materials Parameter Definitions

Use the figures and table below to help you understand the various parameters you can modify for Fluid Materials.

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

Figure 3.96: Fluid materials parameters in the Data Editors panel

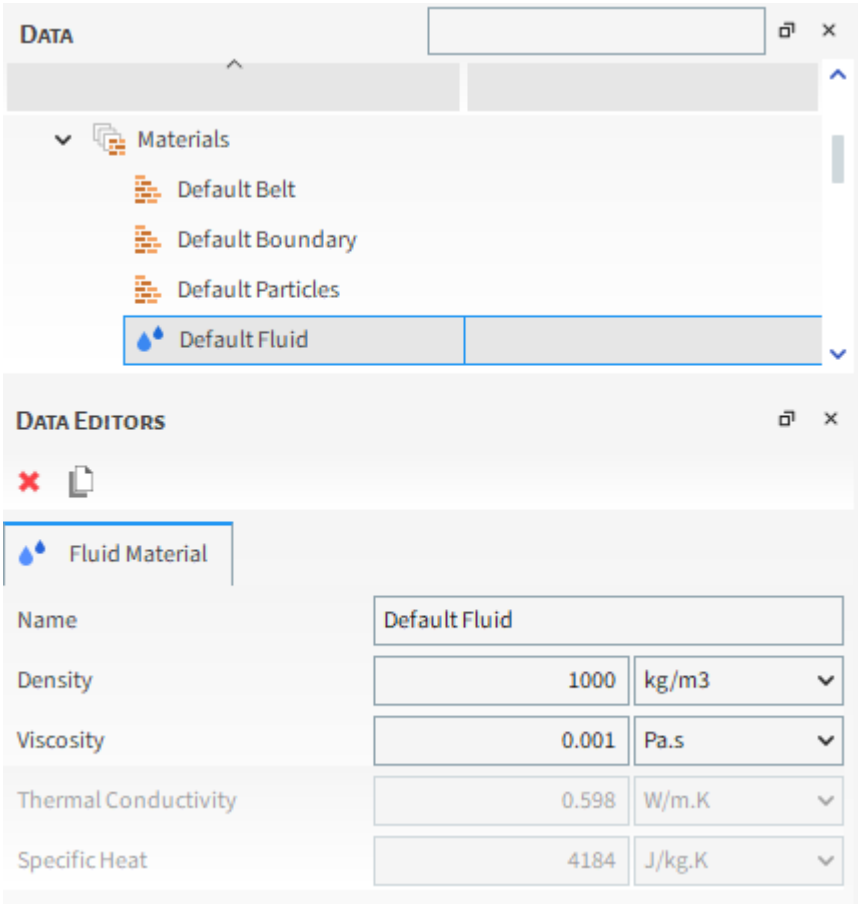


Table 1: Fluid material parameter options

Setting	Description	Range
Name	Enables you to specify a unique identifier for the Material.	99 character limit
Density	The initial density of the SPH elements which is used in the equation of state to compute the relation between pressure and density.	Positive values
Viscosity	Defines the fluid material dynamic viscosity.	Positive values
Thermal Conductivity	When **Thermal Model** is enabled (see also About Physics Parameters (p. 417)), this sets the thermal conductivity value for the material.	Any value
Specific Heat	When **Thermal Model** is enabled (see also About Physics Parameters (p. 417)), this sets the specific heat value for the material.	Any value
Sound Speed*	Parameter used in the equation of state to compute the relation between pressure and density. It also defines the SPH solver time step.	Positive values

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

*While low sound speed can leed to numerical instabilities, high sound speeds increase the simulation cost due to the smaller time step. Refer to SPH technical manual for guidelines on selecting the material sound speed.

See Also:

- Set Simulation Parameters (p. 414)
- Enable Thermal Modeling Calculations (p. 400)

ABOUT MODIFYING MATERIALS INTERACTIONS AND ADHESION VALUES

Rocky enables you to specify unique frictions and adhesion values for each Material-to-Material combination that exists in your simulation, including the interaction of a Material to itself, and also to each other Material defined in the Materials list.

Modify Materials Interactions and Adhesion Values

Modifying Materials Interactions values enables you to calibrate how "wet" or "sticky" a Material acts when it comes into contact with another Material, enabling you to create simulations that more closely mirror real-world conditions.

Tip:

It is good practice to calibrate your particle settings to the real-world behavior of the material you are simulating before changing your materials interaction settings. You may wish to use the Material Wizard and/or the Calibration Suite to help you with this task.

ABOUT MATERIALS INTERACTIONS RELATIONSHIPS

While the Adhesive Force models used for these interaction settings are defined as part of the Set Simulation-Wide Parameters step (see also About Physics Parameters (p. 417)), and the characteristics of the individual Materials are defined in the Modifying Material Compositions step (see also About Modifying Material Compositions (p. 591), the purpose of this step is to define specific material-to-material parameters. These can include parameters for Momentum, Adhesion, and Restitution.

Important:

Even though Rocky enables you to set interactions of geometries to other geometries, these types of interactions have no value to your simulation. The only values that will affect your simulation are those interactions between particles (particle-to-particle) or between particles and boundaries (particle-to-geometry).

MATERIALS INTERACTIONS AND EXTERNAL MODULES

If you have enabled one or more external Modules (p. 1391) that affect your Materials Interactions settings, you might also have separate, module-specific settings that you can define. Refer to the Module's documentation (if provided) for more information. (See also Rocky Simulation Entities that can be Affected by Modules (p. 223).)

MATERIAL INTERACTIONS PARAMETER DEFINITIONS

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

Use the figure and table below to help you understand the various parameters you can modify for Materials Interactions.

Figure 3.97: Materials Interactions parameters in the Data Editors panel for a Constant Adhesive Force

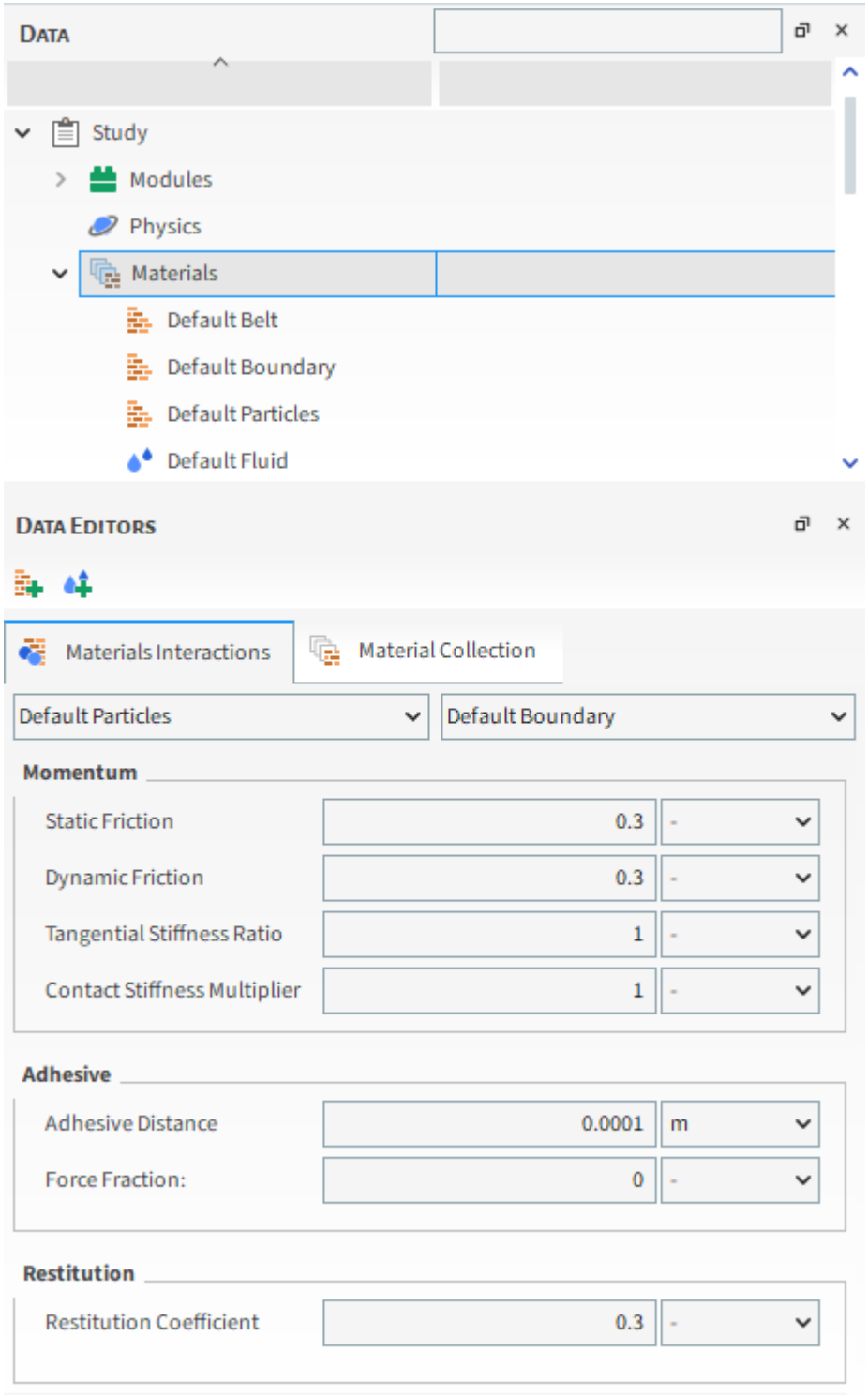


Table 1: Materials Interactions parameter options

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

Setting Momentum	Description	Range
Static Friction	Maximum ratio of contact tangential force to normal force before onset of sliding.	Positive values (see also Best Practices for Setting Up Particles (p. 72))
Dynamic Friction	Maximum ratio of contact tangential force to normal force after onset of sliding.	Positive values; usually less than Static Friction (see also Best Practices for Setting Up Particles (p. 72))
Tangential Stiffness Ratio	When the **Linear Spring Coulomb Limit** model is selected for Tangential Force (see also About Physics Parameters (p. 417)), this defines the ratio of tangential contact stiffness to normal contact stiffness. This parameter controls the bulk Poisson's ratio of granular materials. **Tip:** If knowing the exact relationship between this value and the resulting bulk Poisson's ratio is required, calibration is recommended. This is due to the fact that in addition to this value, other factors including size distribution and friction parameters affect the final bulk Poisson's ratio amount.	0-1 **Tip:** It is recommended that 1 be used for most simulations.
Contact Stiffness Multiplier	When the **Advanced Features** checkbox is enabled on the **Additional Features** tab of the **Options** | **Preferences** dialog (see also About Setting Global Preferences (p. 227)), this enables you to keep (value = 1) or increase (value > 1) the effect of stiffness upon the selected pair of materials when they come into contact during the simulation. Increasing the contact stiffness can help decrease the particle overlap, and is accomplished by multiplying this value by the contact stiffness that Rocky calculates. **Tip:** To learn more about how Rocky calculates Contact Stiffness, refer to the DEM Technical Manual. (From the Rocky **Help** menu, point	1 or greater

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

	to **Manuals** and then select **DEM Technical Manual**.) **Note:** This text field does not support parametric variables. (See also I cannot enter an input variable or mathematical function into a text field (p. 1334))	
Adhesive		
Adhesive Distance	When a Constant or Linear **Adhesive Force** is chosen, this is the distance between the two Materials before forces are applied.	Positive values
Force fraction	When a Constant **Adhesive Force** is chosen, this is the ratio of the adhesive force on the first Material to the gravity force of the second Material.	Positive values
Stiffness fraction	When a Linear **Adhesive Force** is chosen, this is the ratio of adhesive stiffness to the loading elastic stiffness of the contact between the two materials.	0-1
Surface Energy	When a JKR **Adhesive Force** is chosen, this sets what Surface Energy will be used. For more information about these models, see the Rocky Technical Manual. (From the Rocky program **Help** menu, point to **Manuals**, and then click **DEM Technical Manual**.)	Positive values
Restitution		
Restitution Coefficient	Measure of energy dissipation.	Greater than 0 (zero) but less than or equal to 1 **Note:** The recommended range is 0.1 - 1.0
Velocity Limit	When the **Experimental (Beta) Features** checkbox is enabled on the **Options | Preferences** dialog (see also About Setting Global Preferences (p. 227)) and **Velocity Dependent** is selected for **Restitution Model** (see also About Physics Parameters (p. 417)), this is the velocity limit for the restitution.	Values above 10e-5

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

Velocity Exponent	When the **Experimental (Beta) Features** checkbox is enabled on the **Options | Preferences** dialog (see also About Setting Global Preferences (p. 227)) and **Velocity Dependent** is selected for **Restitution Model** (see also About Physics Parameters (p. 417)), this is the velocity exponent for the restitution.	Any value
Other "Modules" Settings		
(Varies)	These settings are specific to only certain *external Module* (p. 1391) and are therefore not documented in the Rocky User Manual. Refer to the Module's documentation (if provided) for more information. (See also Rocky Simulation Entities that can be Affected by Modules (p. 223))	(Varies)

What would you like to do?

- Modify the Interaction or Adhesion Values for a Material-to-Material Set (p. 601)
- Learn more About Physics Parameters (p. 417)
- Learn more About Modifying Material Compositions (p. 591)

See Also:

- Set Simulation Parameters (p. 414)

MODIFY THE INTERACTION OR ADHESION VALUES FOR A MATERIAL-TO-MATERIAL SET

1. Ensure that you have set the **Adhesive Force** parameters you want as part of the Set Simulation-Wide Parameters step. (See also About Physics Parameters (p. 417).)
2. Ensure that you have set the Materials you want and have defined their compositions. (See also About Modifying Material Compositions (p. 591).)
3. From the **Data** panel, select **Materials**. The **Materials Interactions** parameters become active in the **Data Editors** panel.
4. From the **Data Editors** panel, do all of the following:
    a. From **Select Material** list on the left, select the first Material in the Material-to-Material combination for which you want to modify interaction values.
    b. From the **Select Materials** list on the right, select the next Material in the Material-to-Material combination for which you want to modify interaction values. The parameters that define the interaction between the two Materials you selected appear in the **Data Editors** panel.
    c. Modify the values as you want.
    d. Repeat steps 3a-3c for each of the Material-to-Material combinations you want to modify.

See Also:

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

- About Modifying Materials Interactions and Adhesion Values (p. 597)
- About Physics Parameters (p. 417)
- About Modifying Material Compositions (p. 591)

3.2.12. Particle Sets

A Particle set is created when you specify the shape of a particular particle, and then choose the size ranges (and, if desired, the breakage parameters) you want replicated in your simulation.

Particle sets are a subset of the Particles entity (p. 1395), which contain the Properties and Curves for your Particle set(s). You can preview Particle shapes in a Particles Details window, and if you have collected collision statistics during processing, can use that same window to view the results on a representative particle.

See Also:

- Set Simulation Parameters (p. 414)
- Use Properties and Curves (p. 931)

ABOUT ADDING AND EDITING PARTICLE SETS

You can add as many unique Particle sets (p. 1393) containing as many size ranges each to a given simulation as you like, but you must have added at least one Particle set and one Particle Input to process a simulation.

See the sections below for details on the various settings you can assign to a particle set.

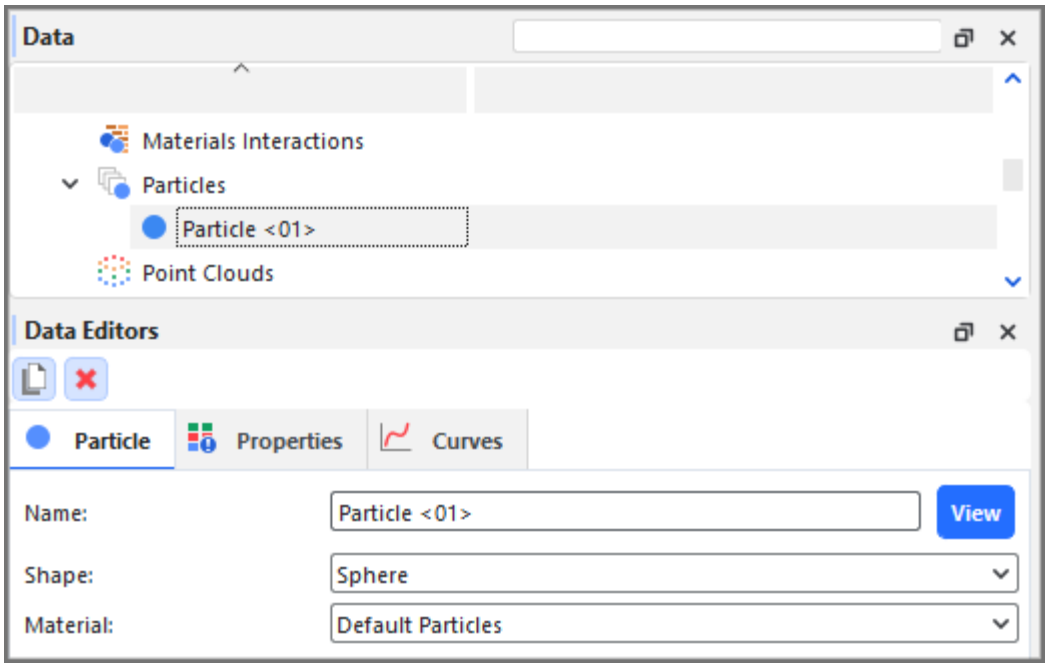
Main Particles Tab and Shape Selection

On the main **Particle** tab (Figure 1), the **Shape** you choose for your Particle set largely determines what models and settings can be applied thereafter.

Shape attributes such as whether your particle is spherical (p. 1395) or shaped (p. 1395); has sharp corners (p. 1395) or rounded corners (p. 1394); made of three-, two-, or one-dimensional components; is convex or concave; and so on, affect whether a particle can be divided (or meshed (p. 1392)) into various Elements (p. 1390), made to be flexible, have calculations such as breakage or thermal applied to them, and more.

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

Figure 3.98: Main Particle tab on the Data Editors panel



There are currently four categories of particle shapes that can be used in Rocky:

- **Fiber** particles, which are strand-like 3D objects made by stringing together linear Segments (p. 1394) and where each segment can have its own thickness. This thickness along with the sieve size (see the Size Tab section below) helps determine the number of elements used to make up the shape (see the Composition Tab section below). Rocky comes with a **Straight Fiber** by default (Table 1), but other **Custom Fiber** shapes (Table 2) can be imported with a text or spreadsheet file. (To learn more about creating a Custom Fiber definition file, see About Defining and Importing Custom Particle Shapes (p. 670).)

- **Shell** particles, which are sheet- or container-like 3D objects made by joining two-dimensional, open surfaces of a definable thickness. Because they contain no interior volume, the mass of a Shell particle is much lower than a similarly shaped Solid (p. 1395) particle. Similar to a Custom Fiber shape, you can determine the thickness of the surfaces that make up the Shell shape which, along with sieve size, affect the mass and the number of elements Rocky will use to make up the shape (see the Composition Tab section below). No default Shell particles come with Rocky, but you can add your own **Custom Shell** shapes (Table 2) by importing an .stl file.

- **Solid** particles, which are closed 3D objects of full interior volume. Seven Solid shapes are provided by default (Table 1), and you can also import your own **Custom Polyhedron** shape (Table 2) from an .stl file. (See also About Defining and Importing Custom Particle Shapes (p. 670).)

- **Assembly** particles, which are made by combining the shape of other Particles sets together into a single, unique particle (Table 3). When compared to importing a similar custom .stl shape, an

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

Assembly (p. 1389) particle has the ability to run much faster due to simpler calculations, especially when multiple convex shapes are assembled to represent a concave shape.

Note:

Different shapes types have different capabilities in Rocky. Refer to Rocky Particle Shapes and Capabilities (p. 41) for specifics.

Table 1: Default shape types in Rocky

Default Spherical (Solid)	Default Shaped (Solid)	Default Fiber
		
Sphere	**Sphero-Cylinder**	**Straight Fiber**
		
	Sphero-Polygon	
		
	Sphero-Polyhedron	
		
	Polyhedron	
		
	Briquette	

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

Default Spherical (Solid)	Default Shaped (Solid)	Default Fiber
	 Faceted Cylinder	

Table 2: Example custom (imported) shape types

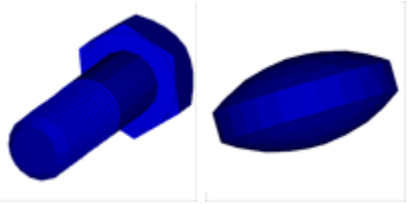
Custom Polyhedron (Solid)	Custom Shell	Custom Fiber
 Examples of a *Concave* (p. 1390) bolt shape and a *Convex* (p. 1390) tablet shape, each imported as a Custom Polyhedron. **Note:** If any portion of a Custom Polyhedron is concave, no matter how small a portion, Rocky considers the whole shape to be concave.	 Example of an snack chip shape imported as a Custom Shell.	 Example of a three-segmented shape imported as a Custom Fiber.

Table 3: Example Assembly shape types

Using Only Default (Solid) Shapes	Using Solid and Straight Fiber Shapes	Using Custom Imported Convex Shapes
 Example of a snowman Assembly shape made from multiple default Solid shapes, including two Spheres, a Sphero-Polygon, and a Faceted Cylinder.	 Example of a lollipop Assembly shape made from a default Sphere and a Straight Fiber shape.	 Example of two custom convex shapes being combined into a concave bolt Assembly shape.

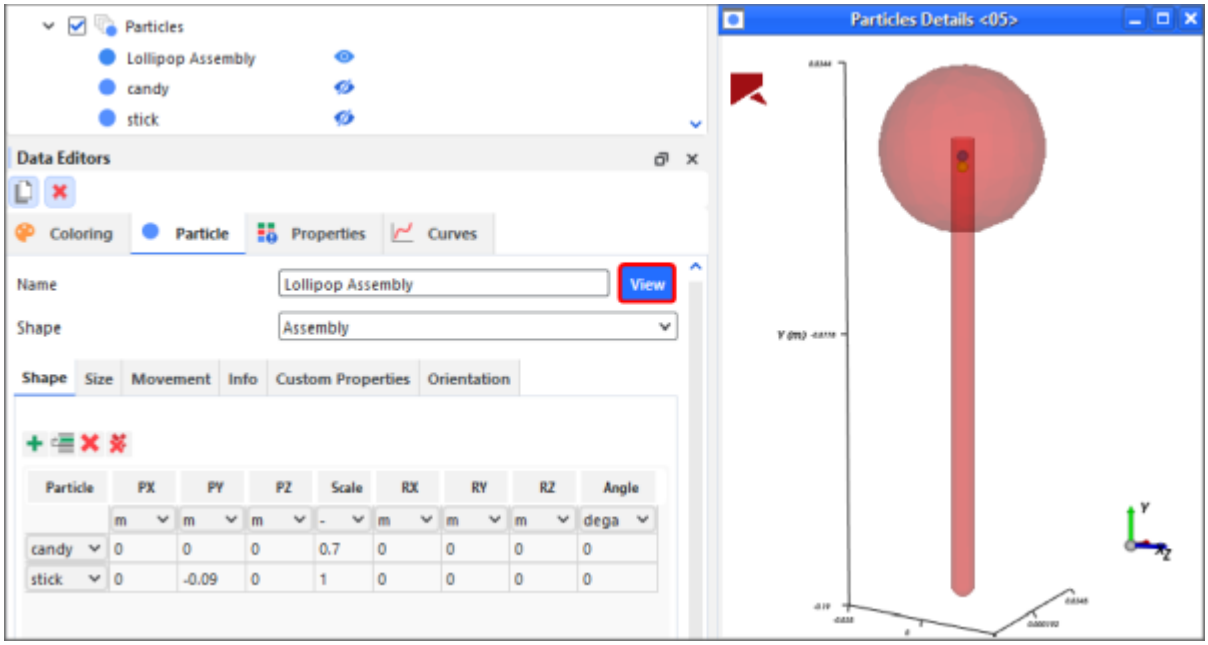
(See also About Particle Assemblies (p. 663).)

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

PARTICLES DETAILS WINDOW

You can review the shape you select (or import) from the **Shape** list by choosing to see a **View** (Figure 2) of the particle shape, which opens in its own window called a **Particles Details** window. (See also About Particles Details Windows (p. 679)).

Figure 3.99: Particle, Shape tab on the Data Editors panel showing a Particles Details window



In this version, the preview reflects the size of the largest particle in the selected Particle set. (See **Size Tab** section below for more information.)

Notes:

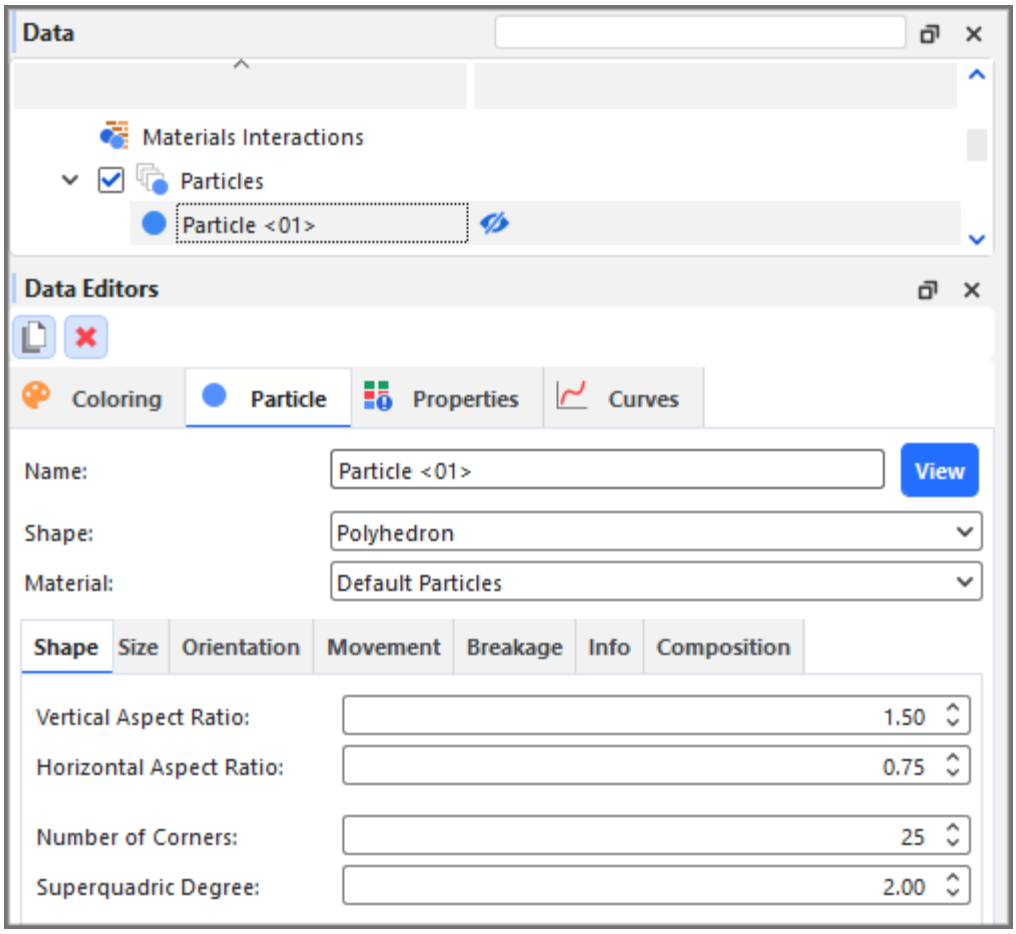
- The **Particles Details** window can also be used to display collision statistics for certain particle shapes after processing. (See also About Intra-Particle Collision Statistics (p. 324).)
- For Assembly (p. 1389) shapes, the **Particles Details** 3D view shows the actual position of the part when the size type is Original Size Scale, and it is set to 1 (which is the default setting).
- The Geometric Center for an Assembly particle is represented in the Particles Details window as a yellow dot, and the Center of Mass is represented by a blue dot.

SHAPE SUB-TAB (DEFAULT SHAPES)

You can modify some of the default shapes that come with Rocky by adjusting the attributes-such as the vertical and horizontal aspects, and number of corners-on the **Shape** sub-tab (Figure 3).

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

Figure 3.100: Particle, Shape tab on the Data Editors panel for a default particle shape



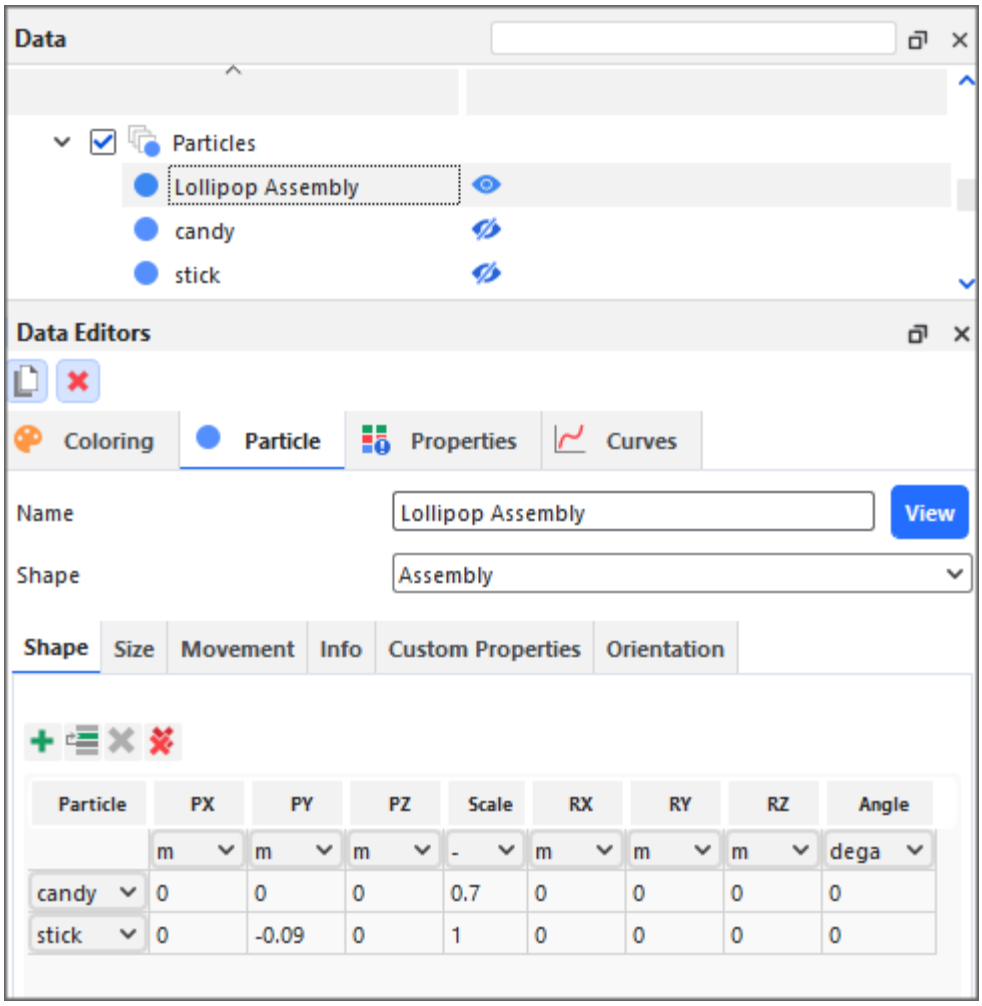
In this way, you can derive a wide variety of different real-particle shapes. (See also Appendix A: Particle Shape Examples (p. 1299).)

SHAPE SUB-TAB (ASSEMBLY SHAPES)

If you are creating an **Assembly** shape, you will have different options on the **Shape** tab (Figure 4). These options enable you to choose which individual particles' shapes (parts (p. 1393)) will make up the final assembled shape, and how those parts are presented (locations, sizes, rotations) in the Assembly. (See also About Particle Assemblies (p. 663).)

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

Figure 3.101: Particle, Shape tab on the Data Editors panel for an Assembly particle shape

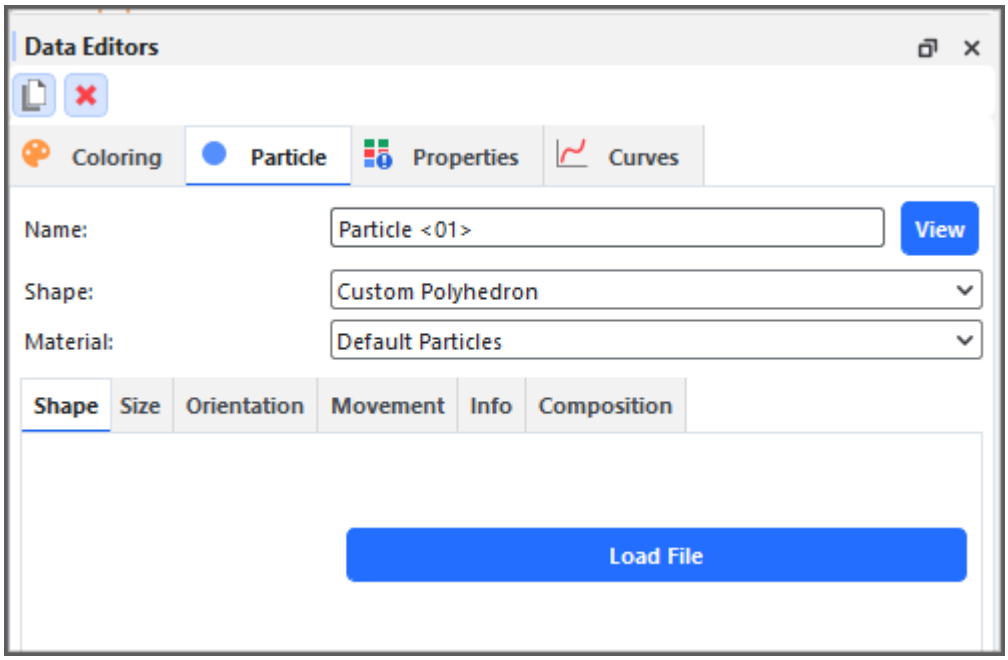


SHAPE SUB-TAB (CUSTOM IMPORTED SHAPES)

After importing your **Custom Polyhedron** shape, you will be able to use the options on the **Shape** tab to select a different STL file (Figure 5).

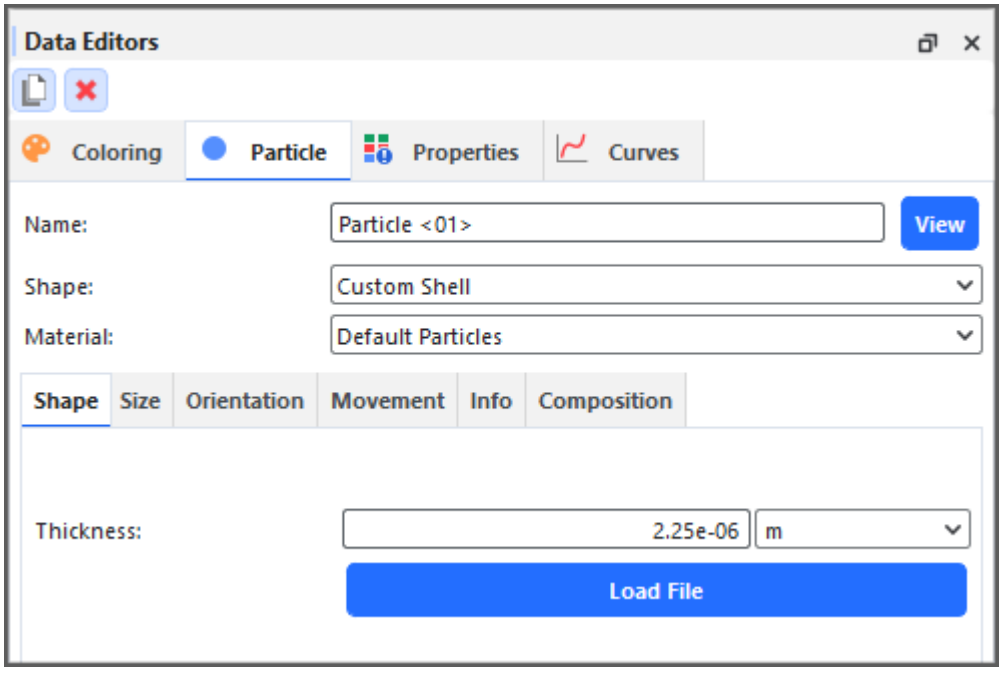
Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

Figure 3.102: Particle, Shape tab after importing a Custom Polyhedron shape



When importing a **Custom Shell** particle shape, Rocky assigns an initial (and arbitrary) thickness value during import. This value is reflected in the **Particles Details** window when 100% **Original Size Scale** is selected on the **Size** tab. (See *Size Tab* section below.) The **Shape** tab will then allow you to enter a new Thickness value so that you may achieve the desired thickness of your imported particle shape (Figure 6).

Figure 3.103: Particle, Shape tab on the data Editors panel after a Custom Shell is imported



The Thickness value that is entered here represents the actual thickness of the largest particle in the set at 100% of its imported size, taking into account any scaling that you have indicated on the **Import**

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

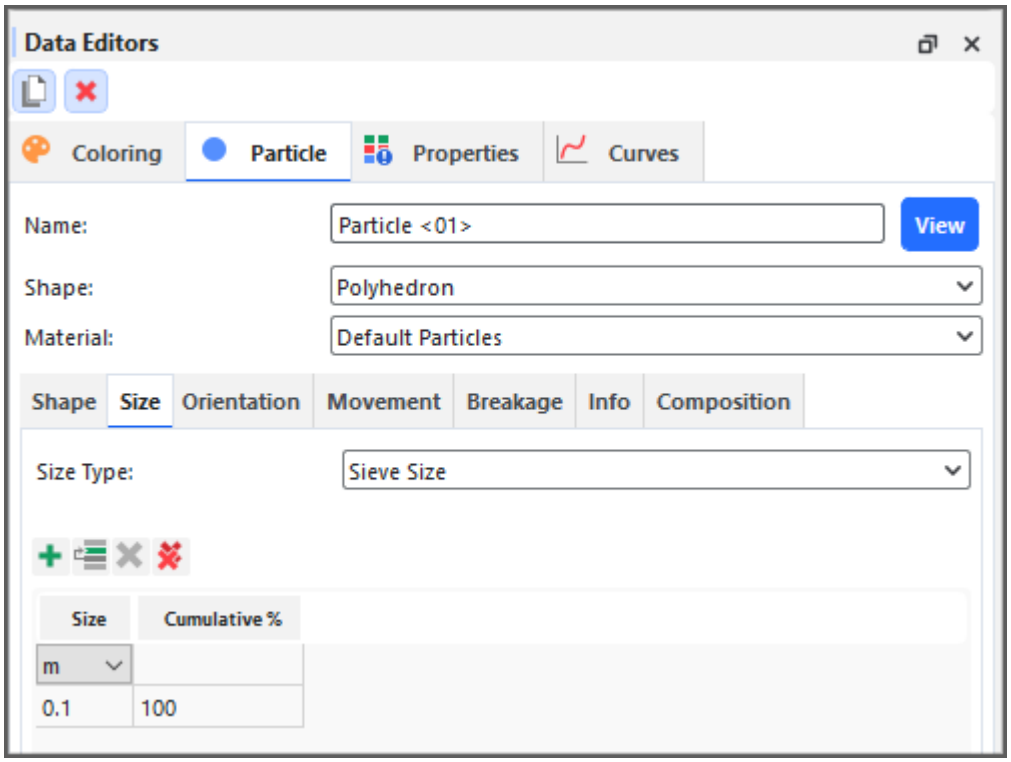
File Info dialog. Halving the Thickness value will reduce the relative thickness of all particles in your set by 50%; doubling the value will increase the relative thickness 200%, and so on.

Once you have set up your particle shape to your liking, you can then choose to export it to an .stl file for further analysis outside of Rocky. (See also Export a Particle Shape to an STL File (p. 698).)

SIZE SUB-TAB

The **Size** tab (Figure 7) is where you define the Particle Size Distribution (PSD) of your Particle set, and the method by which the particle size will be measured.

Figure 3.104: Particle, Size tab on the Data Editors panel



In this version of Rocky, there are three different methods, or **Size Type**s, by which you can define your particle sizes:

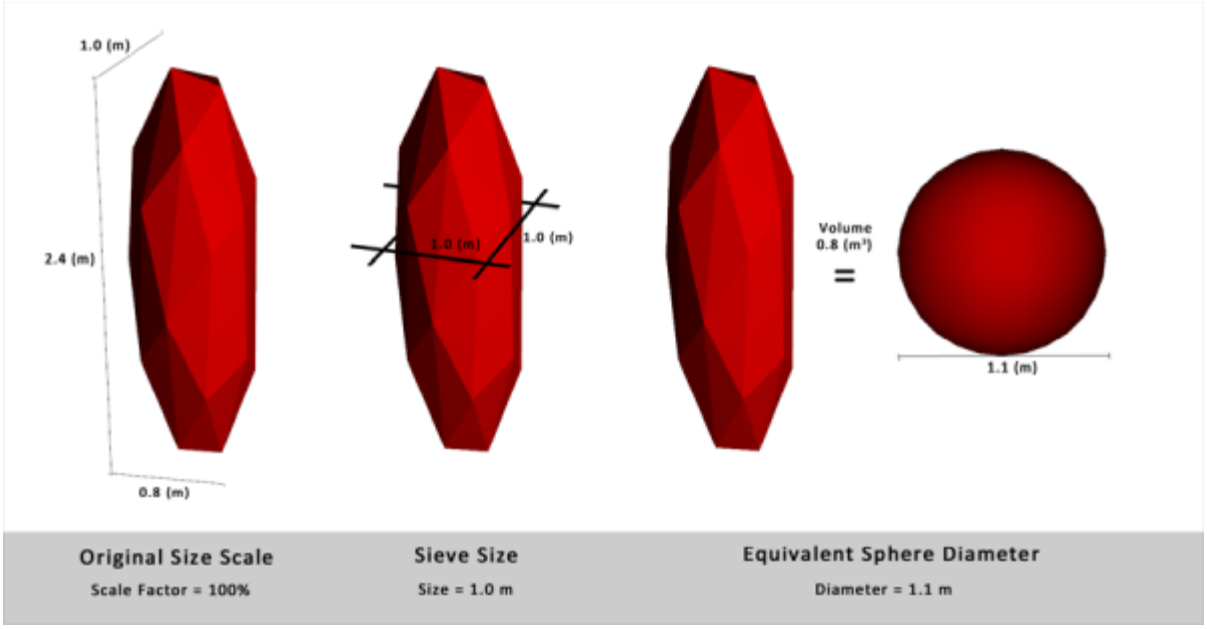
- **Sieve Size** (formerly "Sieve"), which for especially default Rocky shapes, bases particle size upon a virtual mesh

hole just big enough for the particle to pass through. Note: Due to the way Sieve Size is calculated, it should be used only for even and balanced particle shapes (such as the default **Solid** particles included with Rocky) and not for fibers or shapes with high aspect ratios. (For more detail on how Sieve Size is calculated in Rocky, see the DEM Technical Manual. (From the Rocky **Help** menu, point to **Manuals**, and then click **DEM Technical Manual**.)) - **Equivalent Sphere Diameter** (formerly "Equivalent Diameter"), which for especially irregular objects, bases particle size upon the diameter of a sphere with equivalent volume. Tip:To see a walk-through example of setting and using Equivalent Sphere Diameter, refer to the following Workshop: Tutorial - Windshifter in the *Rocky Tutorial Guide*. - **Original Size Scale**, which for especially imported and Assembly shapes, bases particle size upon the original size of the shape. Tip:To see a walk-through example of setting and using Original Size Scale, refer to the following Workshop: Tutorial - Tablet Coater in the *Rocky Tutorial Guide*.

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

The **Size Type** you choose then determines how you reference your resulting PSD: by (sieve) **Size**, (sphere) **Diameter**, or by (original) **Scale Factor** respectively. Figure 8 illustrates the difference between the three types.

Figure 3.105: Comparison of the three different Size Types in Rocky



Note:

The **Particles Details** window for the particle set will reflect the size of only the largest particle in the *Particle set*.

Tip:

To show the exact size of your imported shape within the **Particle Details** window, use the **Original Scale Size** for **Size Type** and set the biggest size to 100%.

The PSD in Rocky is linear by Particle Mass or, on a semi-log scale, by the resulting particle size, and not by number. When you enter the percentage of cumulative mass and then either particle sieve **Size**, sphere **Diameter**, or original size **Scale Factor** (representing the three available **Size Types**, respectively), these represent points on a semi-log plot. Anything between two points follows the linear rule:

$$Mass = A + B * \log(\text{Size})$$
$$Mass = A + B * \log(\text{Diameter})$$
$$Mass = A + B * \log(\text{Scale Factor})$$

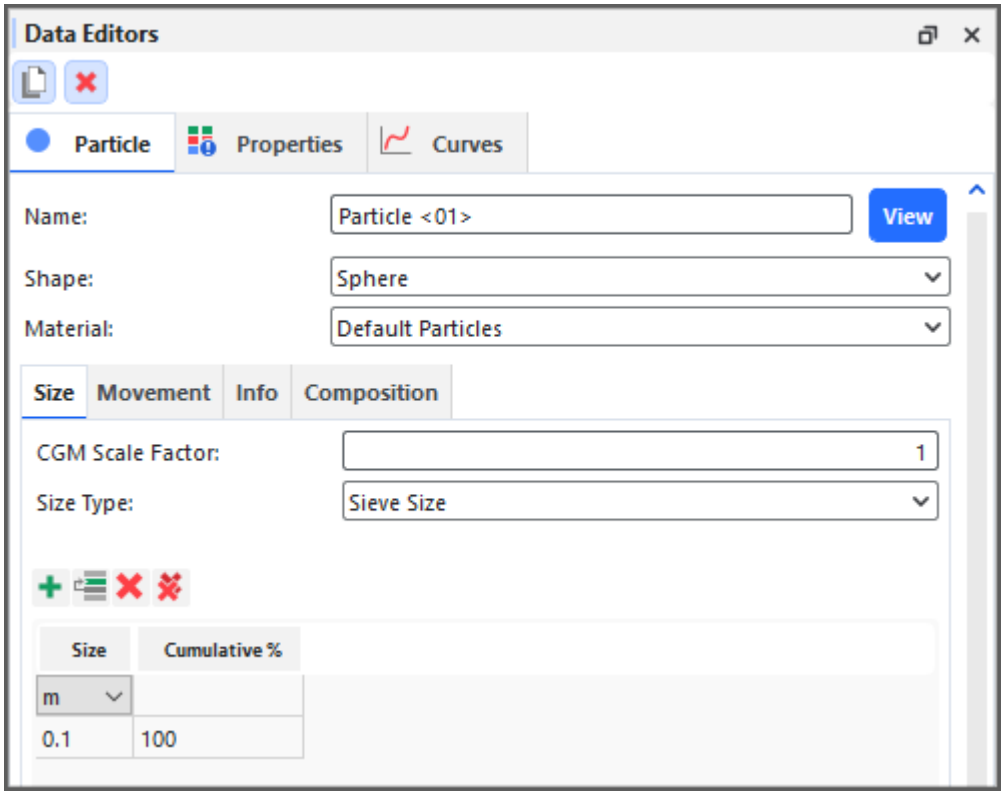
For rigid particles composed of Single Elements, a range of sizes, diameters, or scale factors can be defined for the PSD. For flexible or rigid particles composed of Multiple Elements (also known as meshed (p. 1392) particles-see the **Composition Tab** section below), only one size, diameter, or scale factor should be defined per Particle set.

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

(See also Set Particle Size Ranges (p. 375).)

For simulations with Coarse Grain Modeling enabled (see also About Physics Parameters (p. 417)), there will be an additional option on the Size tab (Figure 9).

Figure 3.106: Particle, Size tab on the Data Editors panel when Coarse Grain Modeling is enabled



The purpose of this additional option is to increase the particle size while maintaining the system behavior using a reduced number of (larger) particles. For example, you can study the behavior of smaller particles-for example, those measured in micrometers or microns (µm)-without having the computational cost of simulating each tiny particle individually. Rather than creating an impractical case involving, perhaps, many thousands of millions of tiny particles, you can analyze the same case but with a reduced number of bigger particles.

This is accomplished by allowing you to specify a unique **CGM Scale Factor** for each of your Particle sets, which enables you to define a much larger, easier-to-process particle whose main purpose is to represent the behavior of a certain amount of smaller particles within your simulation.

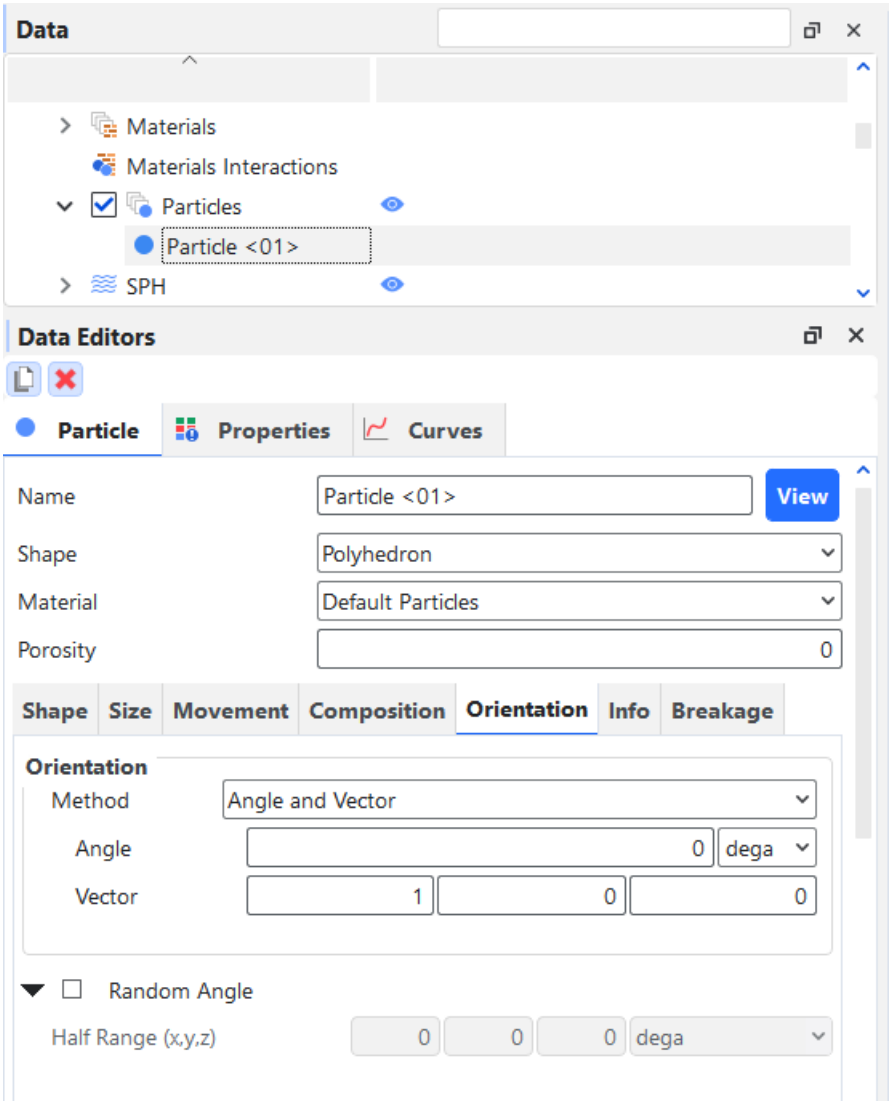
When used, the CGM Scale Factor employs a series of corrections to the larger, representative particle in order to achieve physical behavior coherence with the size of the smaller, real-sized particle. It also ensures that the thermal model, contact model, and various forces used in the simulation will be modified to take into account the characteristics of the smaller, real-sized particle. When a Rocky simulation is carried out with CFD Coupling, the CGM scale Factor will also modify the correlations for drag and thermal properties in a similar manner.

ORIENTATION SUB-TAB

The **Orientation** tab (Figure 10) is where you define what direction your particles will enter the domain.

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

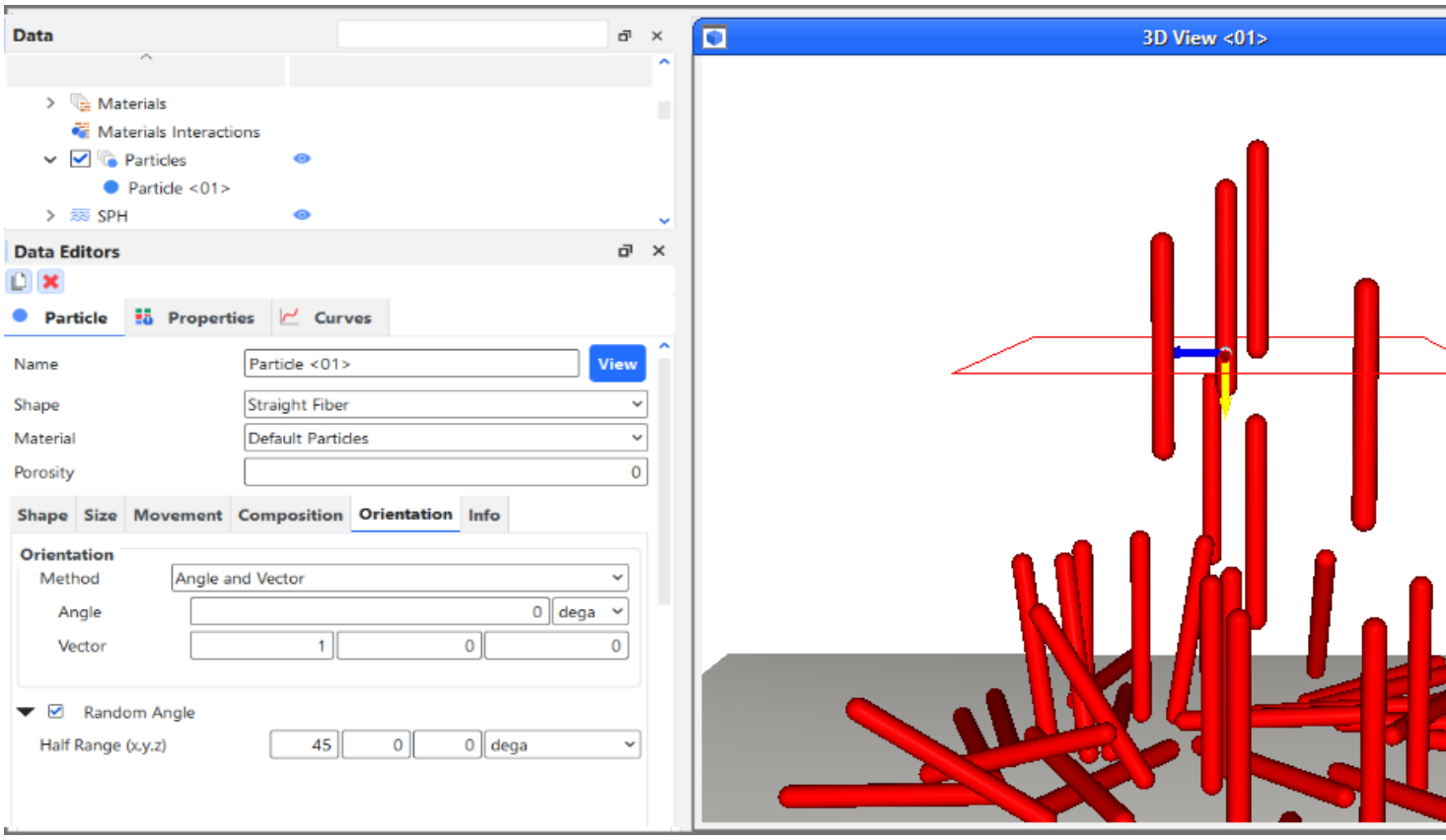
Figure 3.107: Particle, Orientation tab on the Data Editors panel



By default, particles are injected into the domain perpendicular to the inlet (Figure 11). As shown in the examples below, you can change the default orientation in relation to the inlet to have all particles come in at a different angle (Figure 12), have particles come in at random angles (Figure 13), and many options in between.

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

Figure 3.108: A default-orientated Particle set being released from an inlet



Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

Figure 3.109: A Particle set with a 45 degree Rotation Angle along the X axis being released from an inlet

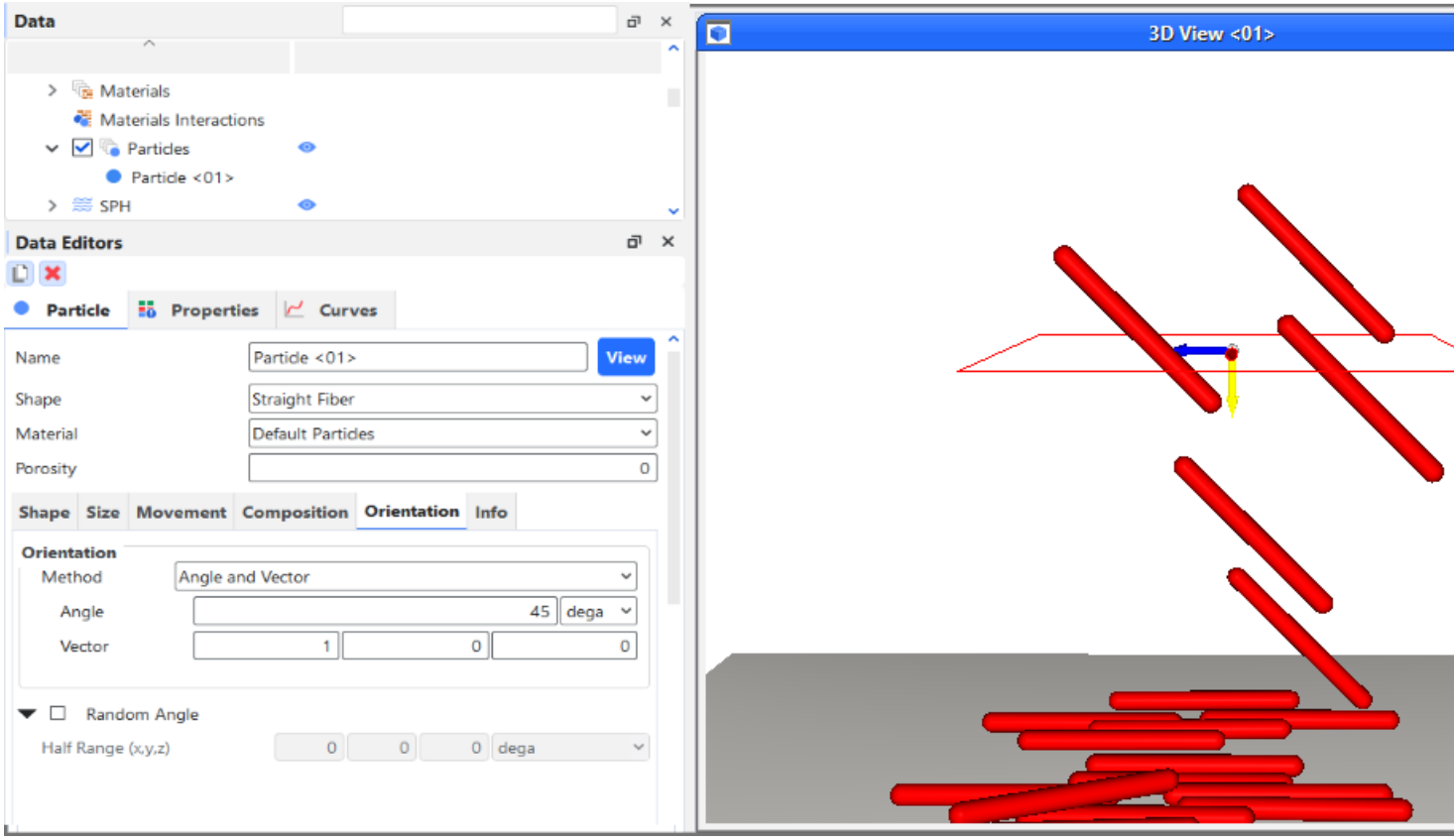
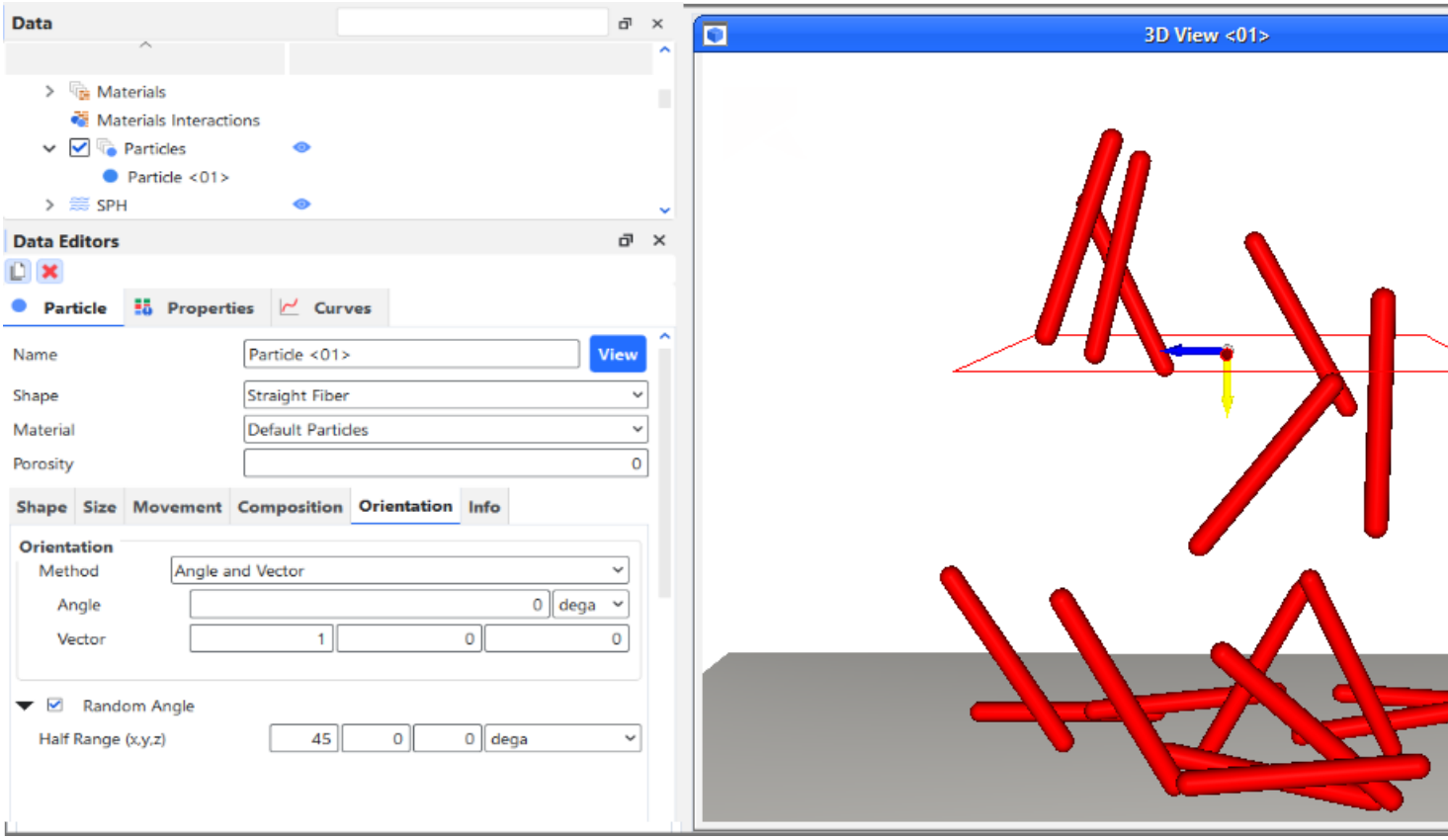


Figure 3.110: A Particle set that allows random orientations within 45 degrees of the X axis being released from an inlet

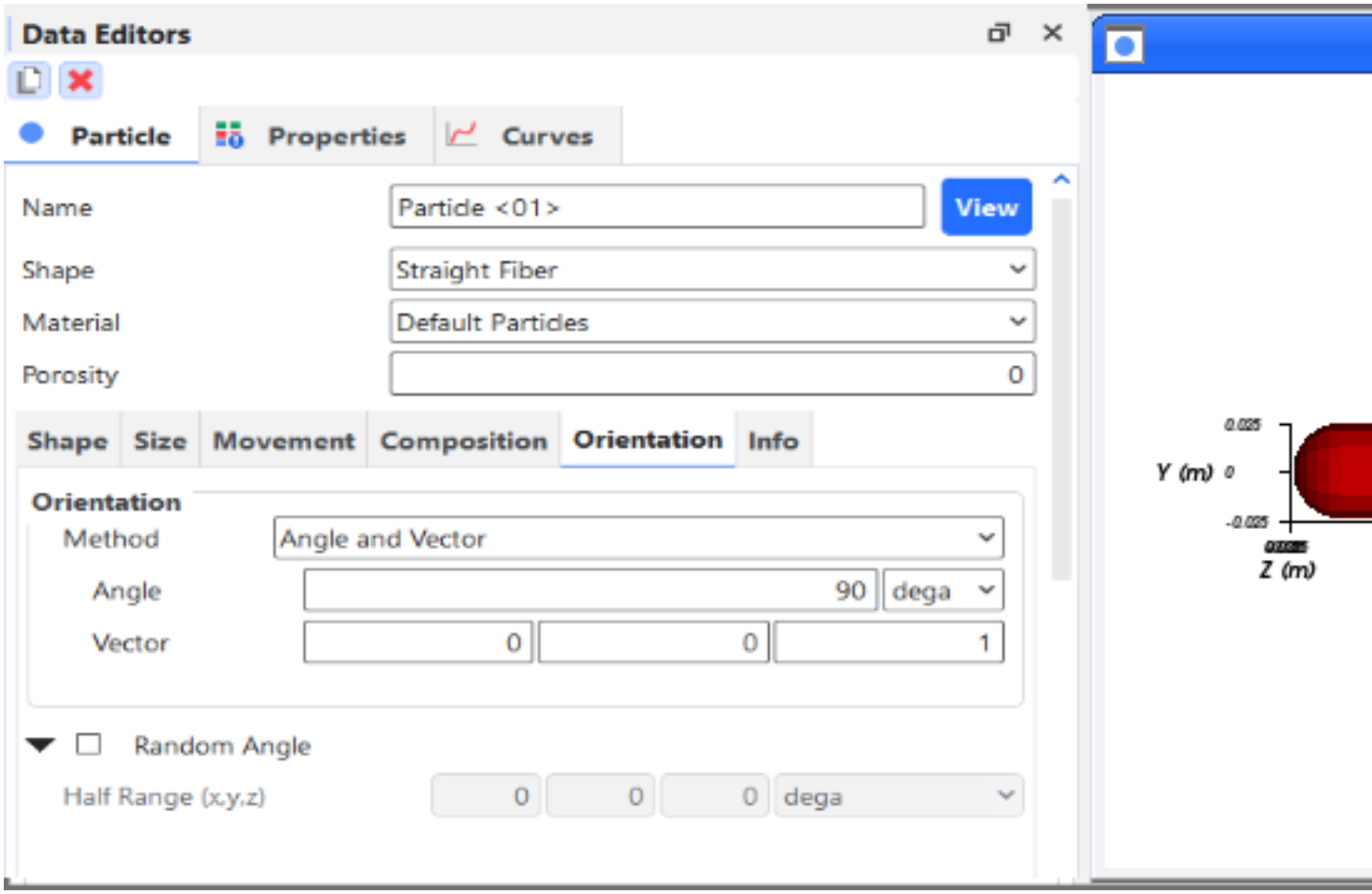


Important: The orientation of the Particle is in relation to the orientation of the inlet from which the Particle set is released. (See also About Adding and Editing Particle Inputs (p. 708).)

Fixed (i.e., not random) particle orientations can be previewed in the Particles Details window (Figure 14). For particles with random orientations, the Particles Details will show only the base orientation off of which the random orientations will be calculated.

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

Figure 3.111: Particles Details window showing example particle rotated 90 degrees in the Z direction

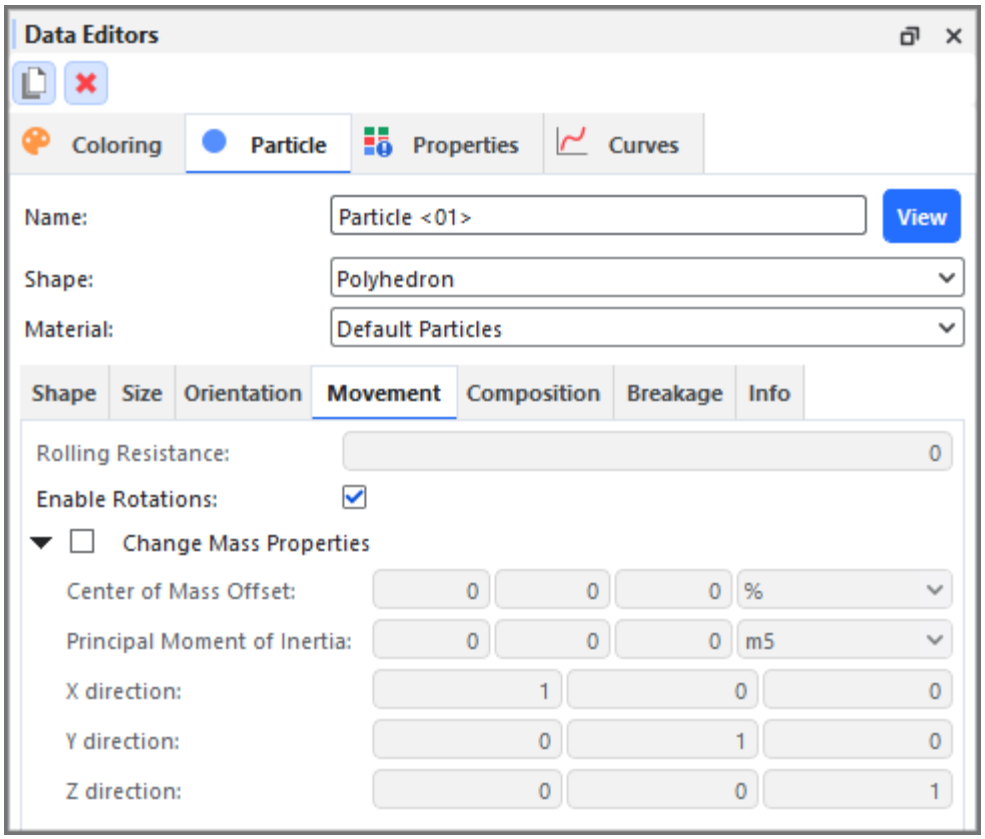


MOVEMENT SUB-TAB

The **Movement** tab includes the rolling resistance, rotations, and (for non-Assembly shapes only) particle mass parameters that you can apply to a Particle set, as shown in Figure 15. For Assembly particle shapes (Figure 15a), the mass properties are calculated automatically by Rocky unless you have decided to override these values by using the **Custom Properties** tab. (See also the *Custom Properties Sub-Tab* section below.)

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

Figure 3.112: Particle, Movement tab on the Data Editors panel



Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

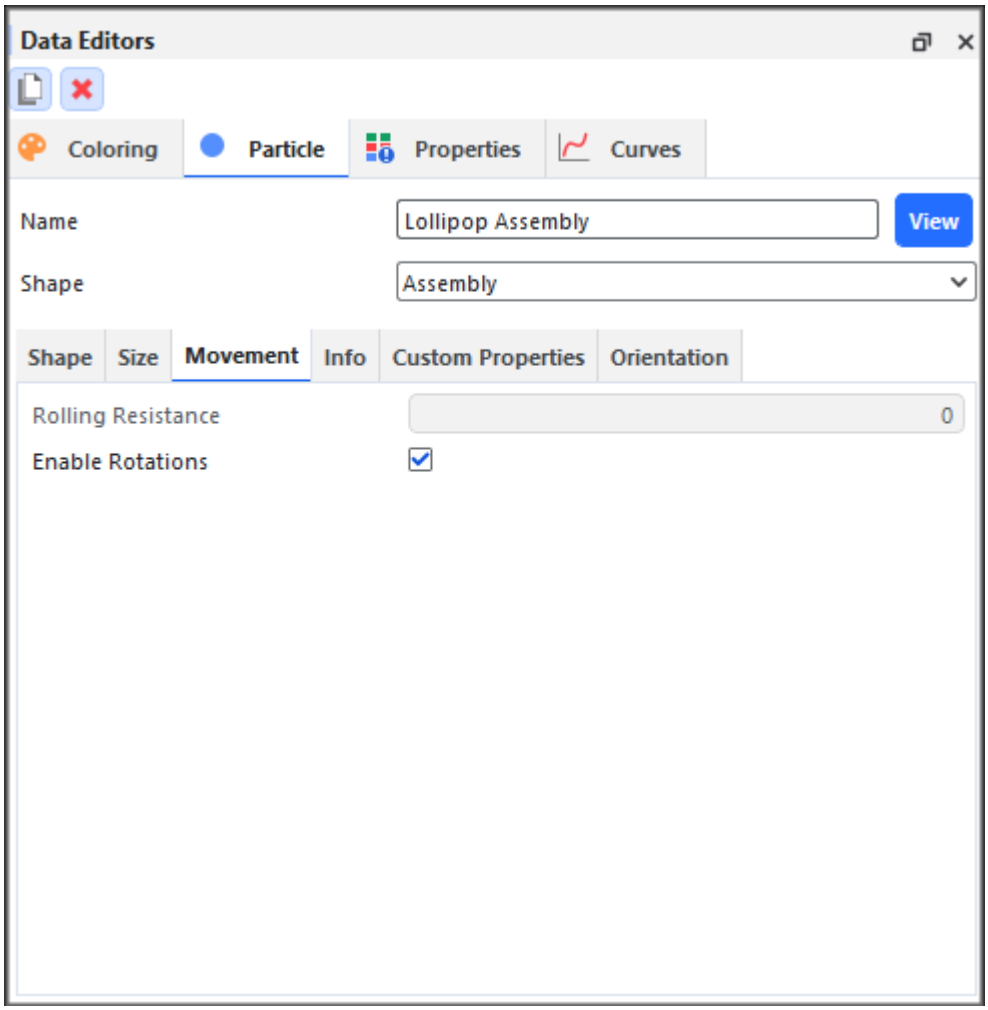


Figure 15a: Particle, Movement tab on the Data Editors panel for an Assembly particle shape

The **Rolling Resistance** value can be set only after a **Rolling Resistance Model** has been defined in the **Physics | Momentum** tab. (See also About Physics Parameters (p. 417).) In addition, Rolling Resistance can be set only for rigid particles that are convex in shape. Therefore, flexible particles and concave-shaped rigid particles will always have zero (0) Rolling Resistance.

Enable Rotations can be edited only for rigid particles. Therefore, flexible particles will always have Enable Rotations turned on. Note: Although flexible particles will always include rotations, you can use the **Include Rotational Deformations** checkbox on the **Breakage** sub-tab if you want to disregard stresses caused by rotational deformations (bending or torsion) when considering *Discrete Breakage* criteria.

While any non-Assembly particle shape can make use of them, Particle mass properties are intended primarily for custom imported particle shapes (see also About Defining and Importing Custom Particle Shapes (p. 670)) and can be used to shift the particle's center of mass and/or override its moment of inertia.

CUSTOM PROPERTIES SUB-TAB

The **Custom Properties** tab (Figure 15b) is for Assembly particle shapes and should be used only if the following criteria are met:

- Two or more parts making up your Assembly particle overlap.

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

- The Rocky-defined Mass, Area, Volume and other properties are not accurate enough for your simulation purposes.

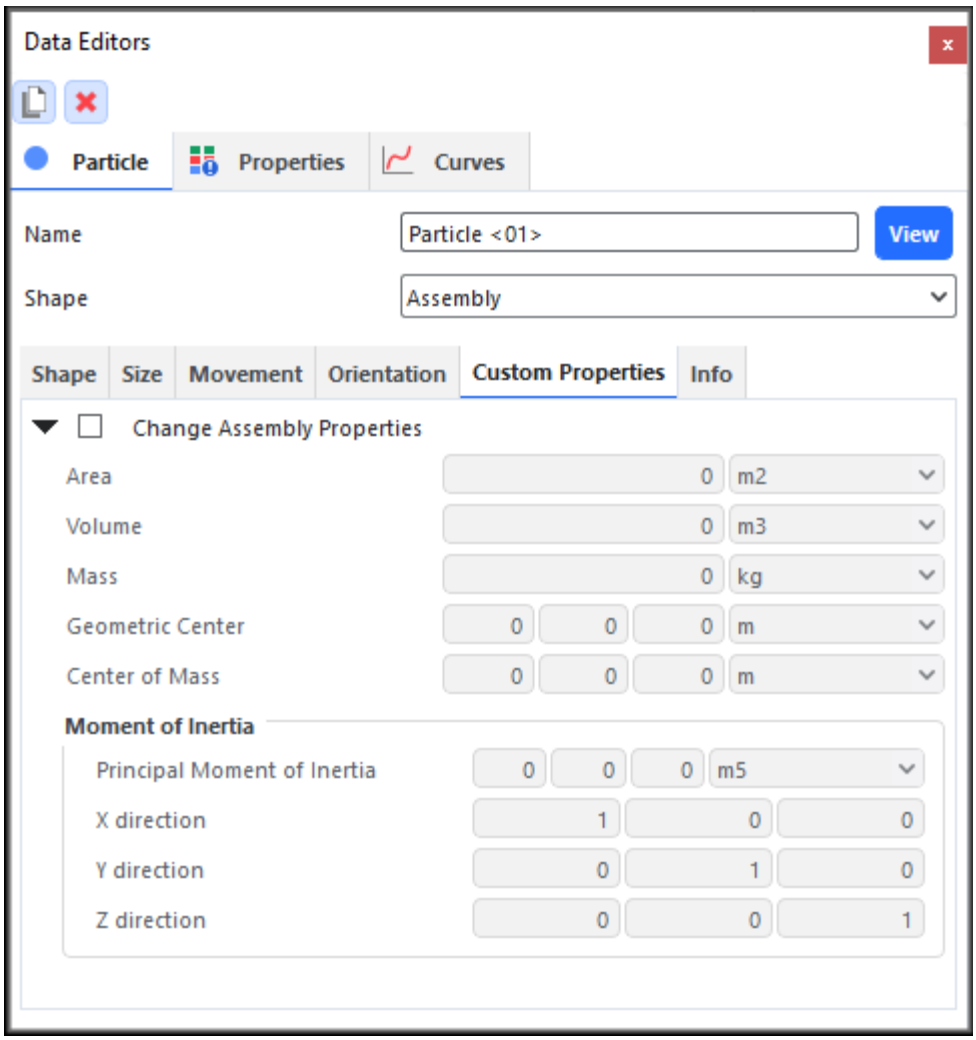


Figure 15b: Particle, Custom Properties tab on the Data Editors panel for Assembly particle shapes

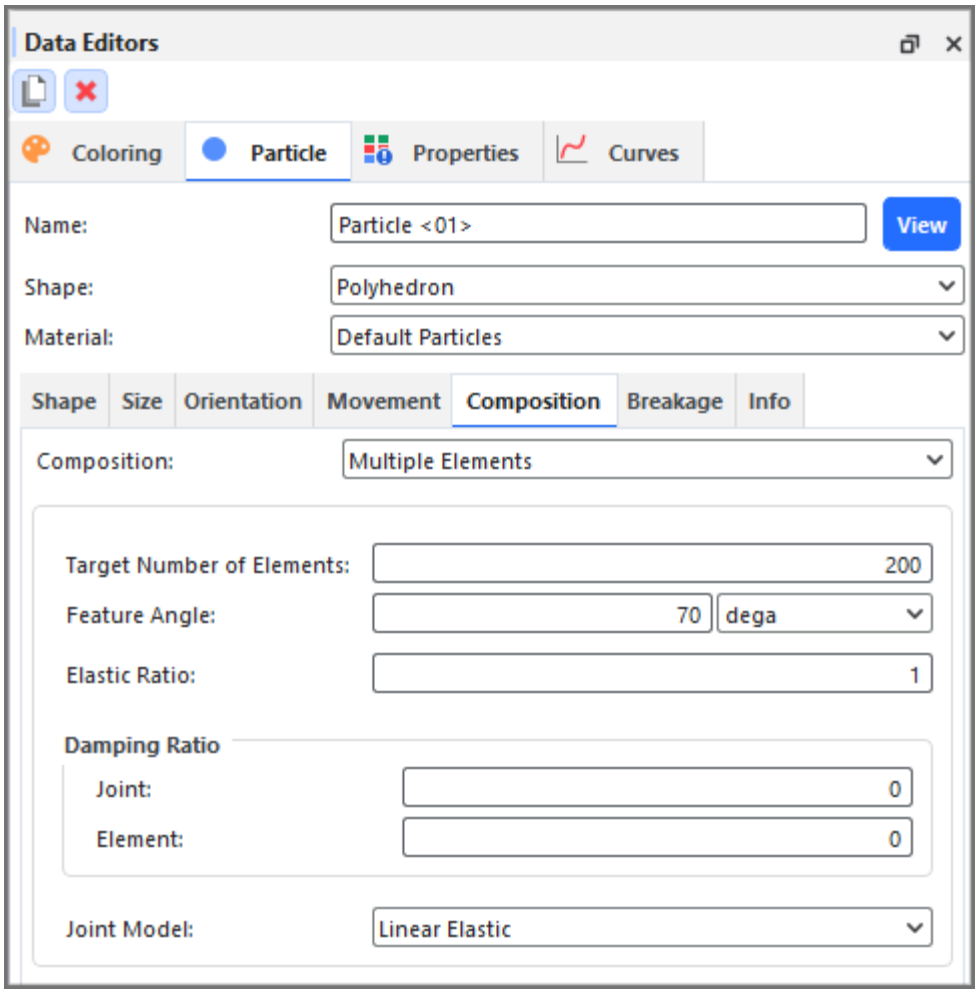
If you decide to define your own Custom Properties for Assembly shape, you can discover the values you need from the CAD program with which you designed your ideal (Assembly) particle shape.

COMPOSITION SUB-TAB

The **Composition** tab contains the meshing (p. 1392) and flexibility and joint (p. 1392) plasticity models that are available for the selected Particle set (Figures 16-17).

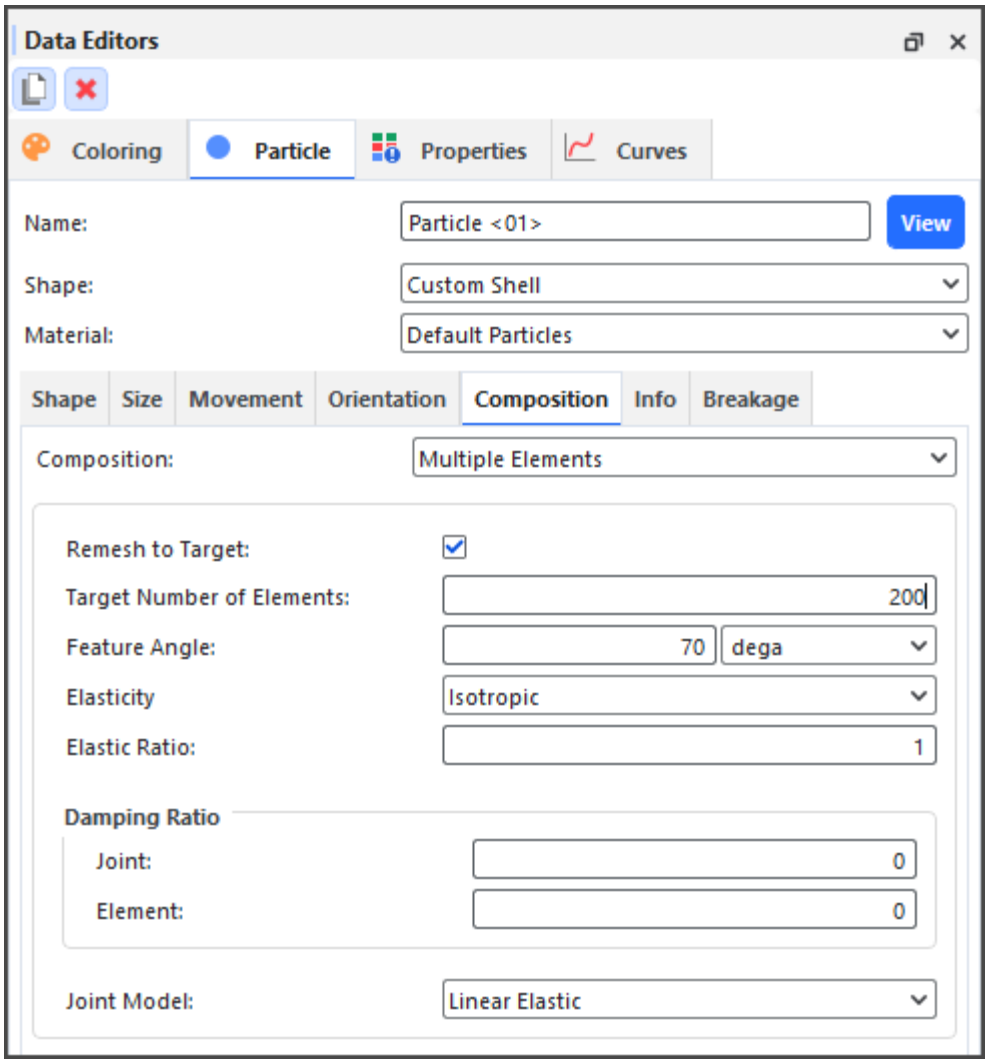
Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

Figure 3.113: Particle, Composition tab on the Data Editors panel



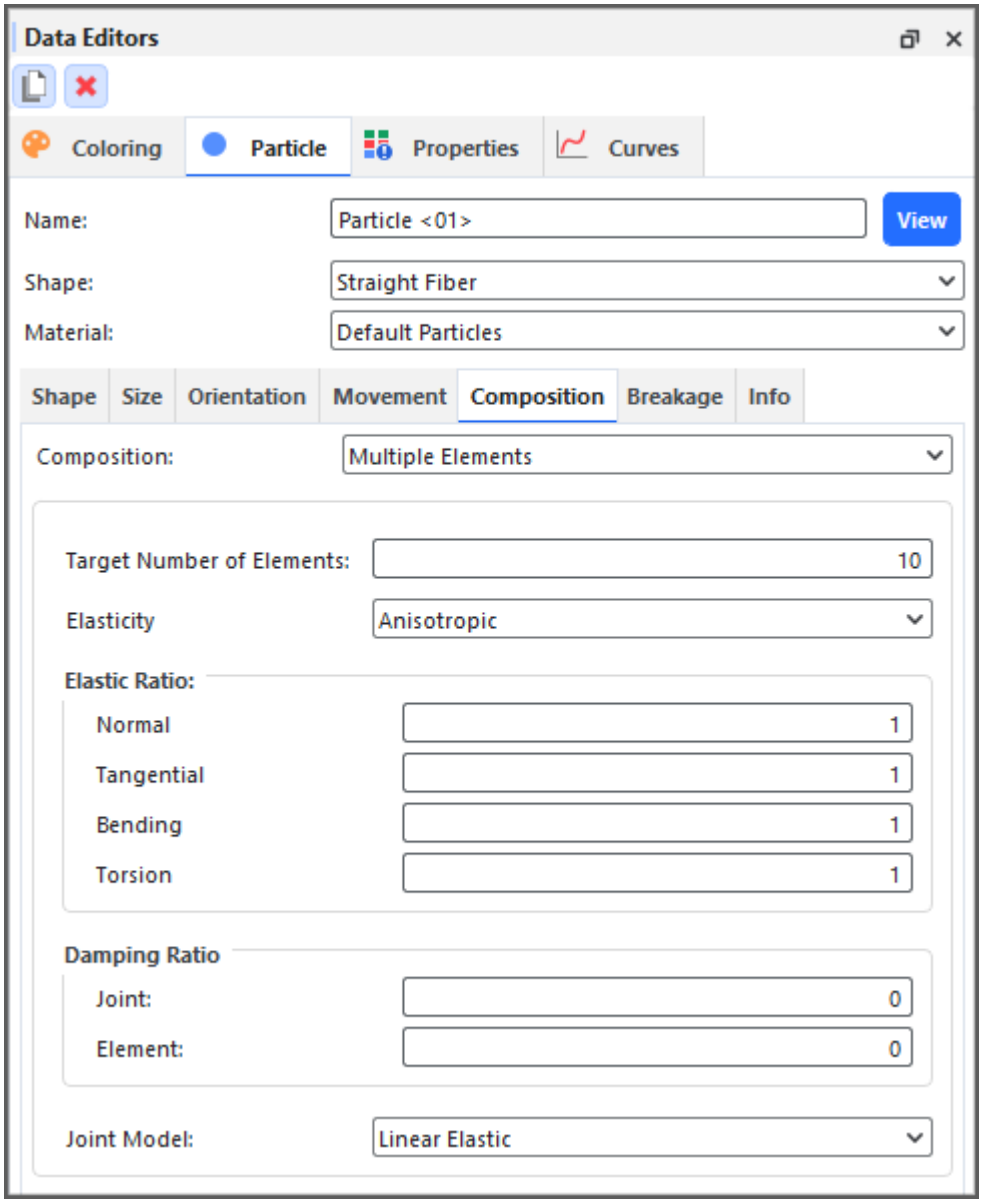
Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

Figure 3.114: Particle, Composition tab on the Data Editors panel for Shell particles



Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

Figure 3.115: Particle, Composition tab for Fibers when both Anisotropic and Bilinear Elastoplastic Joint Model are enabled

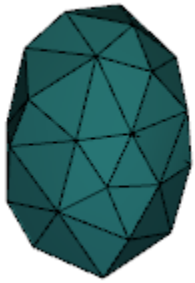
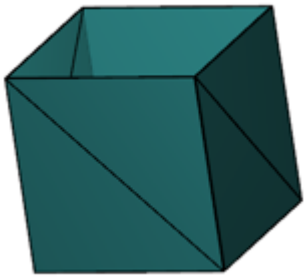


By default, each Rocky particle is generated as a single, undivided, inflexible element. When **Multiple Elements** are selected as the **Composition** however, the particle shape will be composed of many separate Elements (p. 1390). Rocky calculates these separate elements by applying a meshing (p. 1392) model, which enables the particle to become flexible if desired, and in the case of Fibers (p. 1391), can also provide additional detail when coupled with CFD fluid forces.

Meshing is implemented in different ways depending upon which kind of shape you select, as described in Table 4 below.

Table 4: How meshing is implemented in different particle shapes

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

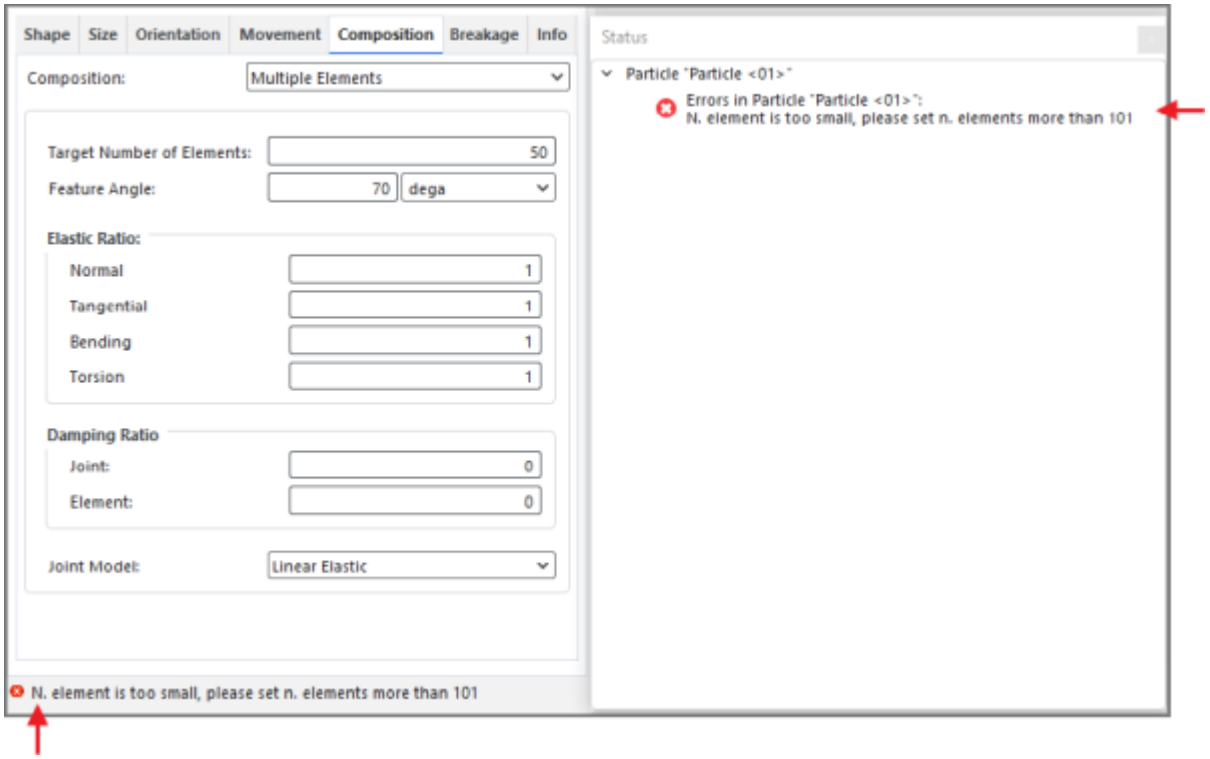
Solid Shapes (3D) (Polyhedron and Custom Polyhedron only)	**Shell Shapes (2D)**	**Fiber Shapes (1D)**
		
When **Multiple Elements** are selected, Solid shapes are divided into a minimum amount of tetrahedrons that are (roughly) equal in volume. The number of tetrahedrons can be (roughly) increased by using the **Target Number of Elements** field, but can only be decreased to the minimum amount of elements calculated by Rocky.	By default when **Multiple Elements** are selected, Shell shapes are divided into a minimum amount of triangles that are (roughly) equal in area. The number of triangles can be (roughly) increased by using the **Target Number of Elements** field, but can only be decreased to the minimum amount of elements calculated by Rocky. When the **Remesh to Target** checkbox is cleared, however, Shell shapes are divided along the same triangular segments of the geometry's imported .stl file. In this situation, the number of triangle elements can only be increased or decreased in the .stl file itself (using a CAD program, such as Ansys SpaceClaim) and then re-imported into Rocky.	When **Multiple Elements** are selected, Fiber shapes are divided into "links" of (roughly) equal length. The number of divisions can be (roughly) increased or decreased by using the **Target Number of Elements** field.

You can choose (roughly) how fine to mesh your particle by entering the **Target Number of Elements** you want. Because each particle shape will have different minimum element amounts, and different element ranges outside of which the meshing cannot maintain the integrity of the selected shape, Rocky helps you set an appropriate value in the following two ways:

- By indicating the minimum number of elements in an error message located at both the bottom of the Data Editors panel and within the Status panel (Figure 19).
- By providing how many actual Elements (or tessellations) the particle will be composed of via the **Elements** value on the **Particle** | **Info** sub-tab. (See also the **Info Tab** section below.)

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

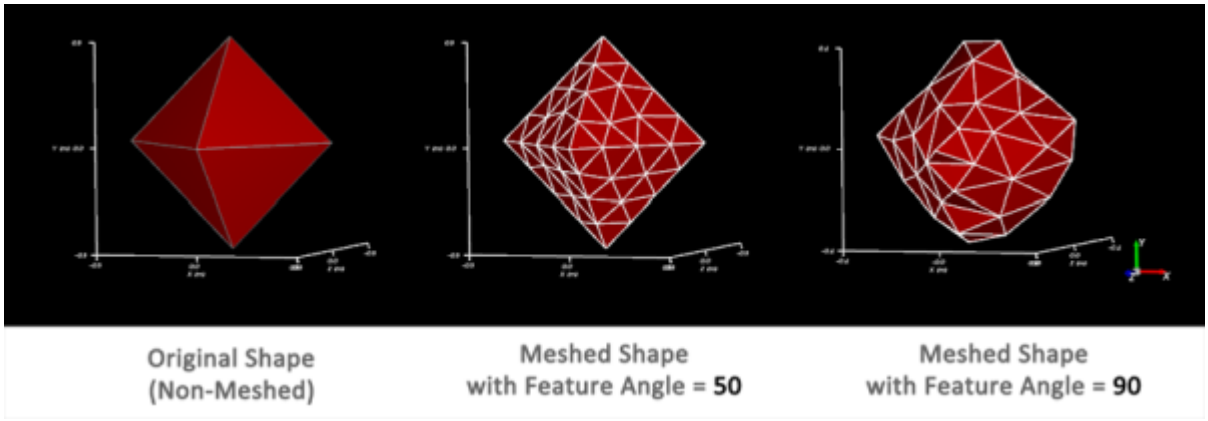
Figure 3.116: Example error messages shown when a Target Number of Elements value is below the minimum calculated by Rocky



Besides (roughly) controlling the quantity of the individual Elements, for Solid and Shell particles Rocky also allows you to (roughly) control the shape of the individual Elements that make up the meshed particle. In cases where the original particle is made up of triangles with sharp edges, the meshing process can change the angles of those edges resulting in a final particle shape that can differ-perhaps undesirably-from the original. It is for this reason that Rocky provides an **Feature Angle** parameter (Figure 20) that enables to you specify how much of the original triangular edges are preserved during meshing, and how many are ignored. In this way, you can have more control over the final shape of your meshed particle, while still retaining its ability to become flexible.

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

Figure 3.117: Original shape compared with its meshed version with different Feature Angle settings



Note:

The value you set for Feature Angle can influence the minimum amount of Elements required by Rocky.

Tip:

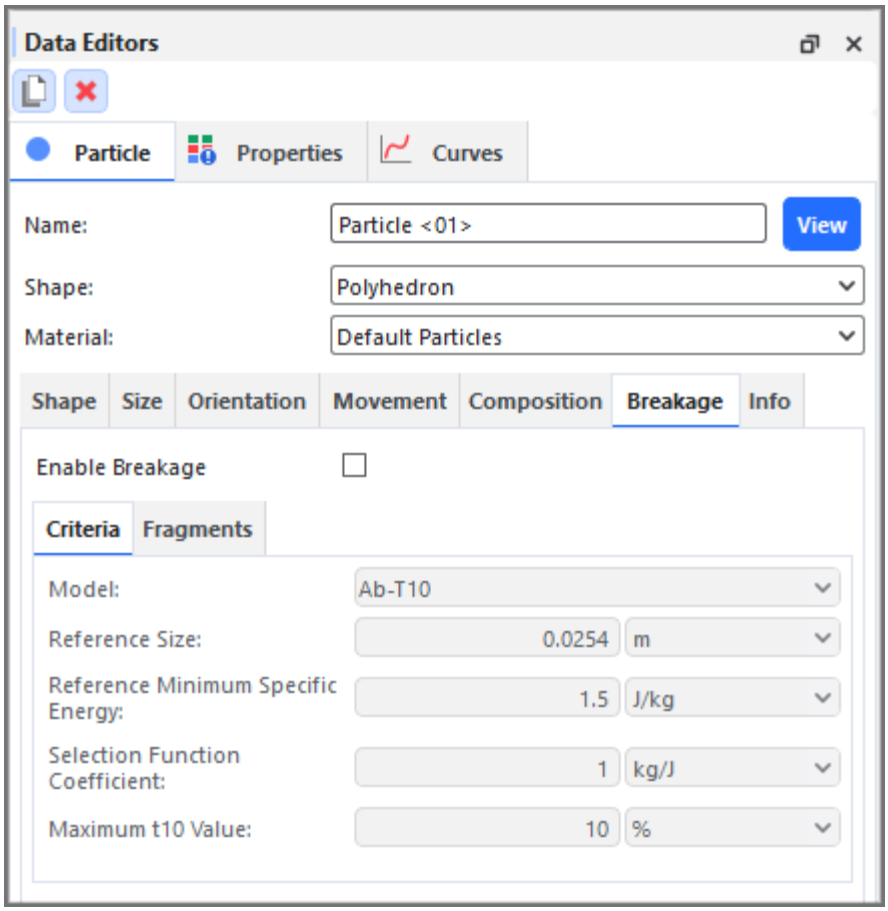
You can visualize the individual Elements in the Particles Details window by turning on **Edges** from the Coloring tab.

BREAKAGE SUB-TAB

The **Breakage** tab settings determine how the particles break and fragment upon contact with other particles or boundaries (Figure 21).

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

Figure 3.118: Particle, Breakage tab on the Data Editors panel



Both the **Shape** and the **Composition** of your particle set will determine whether or not breakage modeling is available, and whether Instantaneous Breakage (p. 1392) or Discrete Breakage (p. 1390) models can be applied. (See also About Particle Breakage (p. 665).)

In this version of Rocky, only the following **Shape** | **Composition** option combinations support breakage calculations (Table 5).

Table 5: Shape Type and Composition Combinations that Support Breakage Calculations

Shape Type	Composition
Either Straight or Custom *Fiber* (p. 1391)	Multiple Element
Custom *Shell* (p. 1395)	Multiple Element
Solid (p. 1395) Polyhedron	Either Single or Multiple Element
Solid Briquette	Single Element
Solid Faceted Cylinder	Single Element
Solid Custom Convex Polyhedron	Either Single or Multiple Element
Solid Custom Concave Polyhedron	Multiple Element

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

Once you choose a compatible particle shape and composition, there will be two kinds of breakage models available: one set of Instantaneous Breakage (p. 1392) models for rigid particles composed of only a Single *Element* (p. 1390), and another set of Discrete Breakage (p. 1390) models that are specific for flexible particles composed of Multiple Elements.

For single-element particles, the Instantaneous Breakage models available include the **Ab-T10** and the **Tavares** models. Both models preserve both mass and volume.

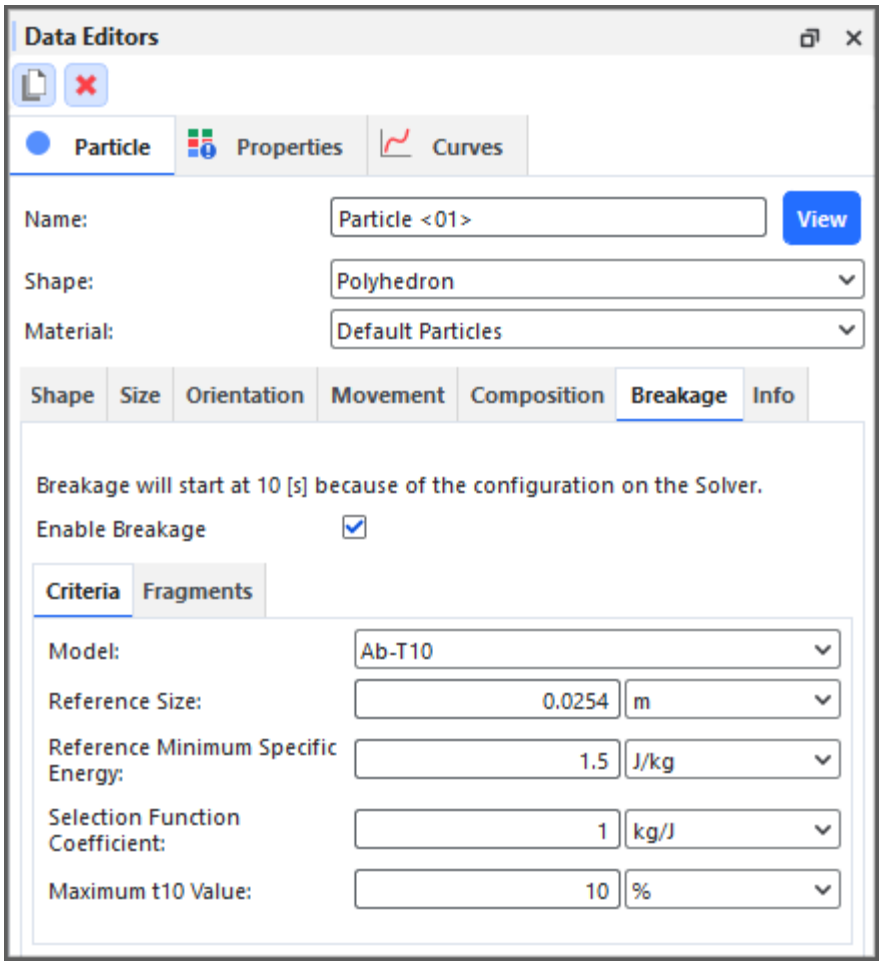
Note:

In this version of Rocky, the **TBM** breakage model is provided as an external module (p. 1391). Refer to the Install an External Module (p. 223) topic for details.

For multi-element (flexible) particles, the Discrete Breakage models available include **Griffith Energy Criterion**, **Shear Stress Criterion**, **Tensile Stress Criterion**, **Tensile or Shear Stress Criterion**, and **von Mises Stress Criterion**.

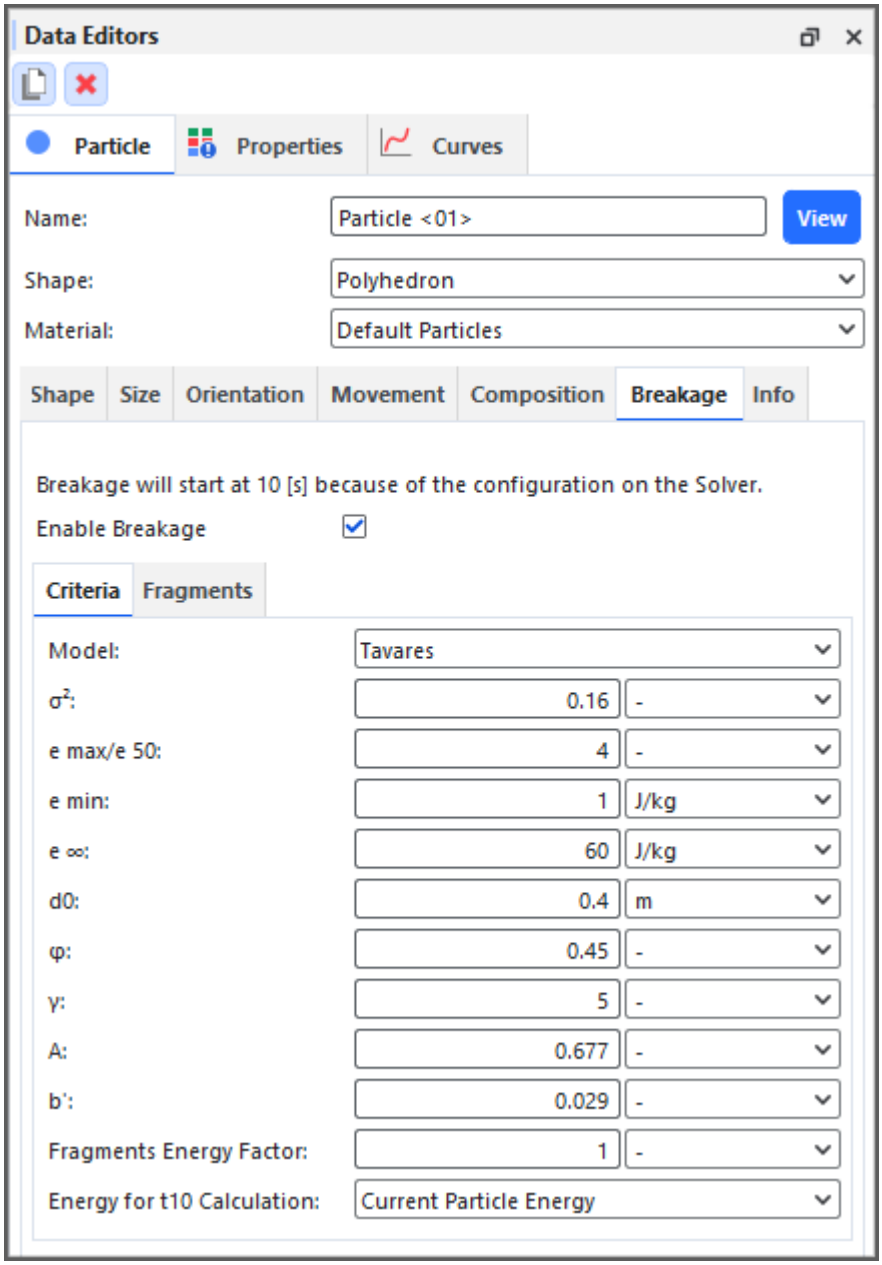
Depending upon which breakage **Model** you choose, different settings will be provided (Figures 22-28).

Figure 3.119: Particle, Breakage tab on the Data Editors panel showing Ab-T10 model settings



Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

Figure 3.120: Particle, Breakage tab on the Data Editors panel showing the Tavares model settings



Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

Figure 3.121: Particle, Breakage tab on the Data Editors panel showing the Griffith Energy Criterion model settings

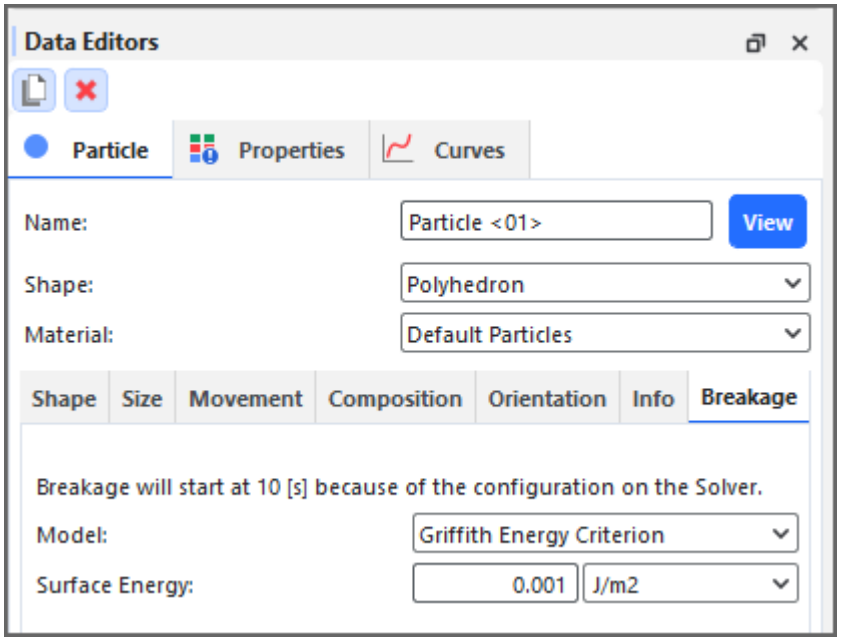
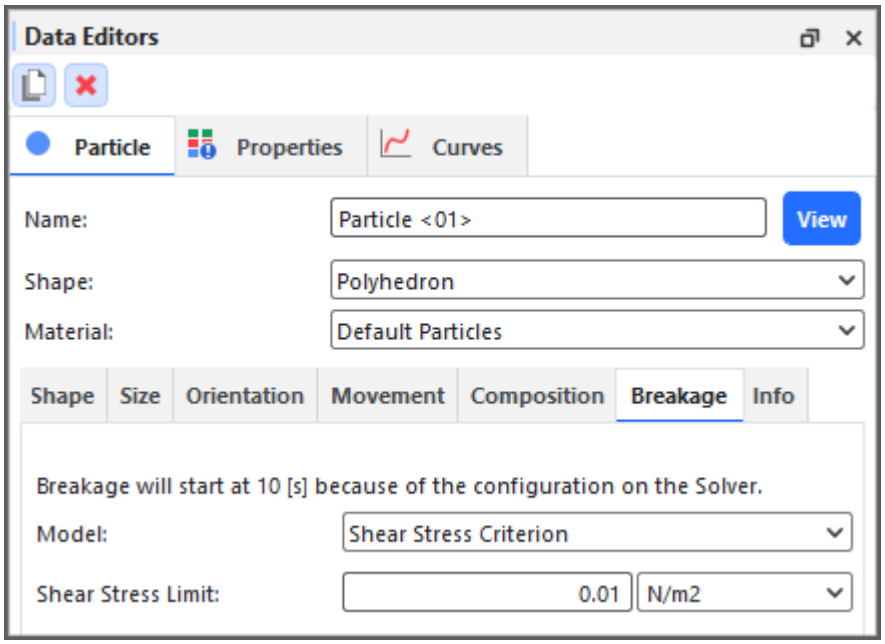


Figure 3.122: Particle, Breakage tab on the Data Editors panel showing the Shear Stress Criterion model settings



Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

Figure 3.123: Particle, Breakage tab on the Data Editors panel showing the Tensile Stress Criterion model settings

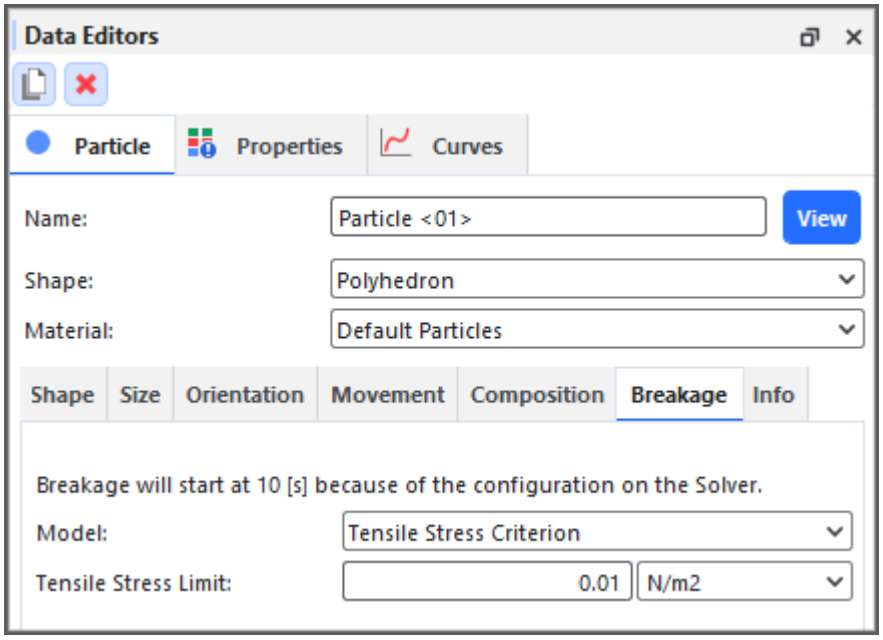
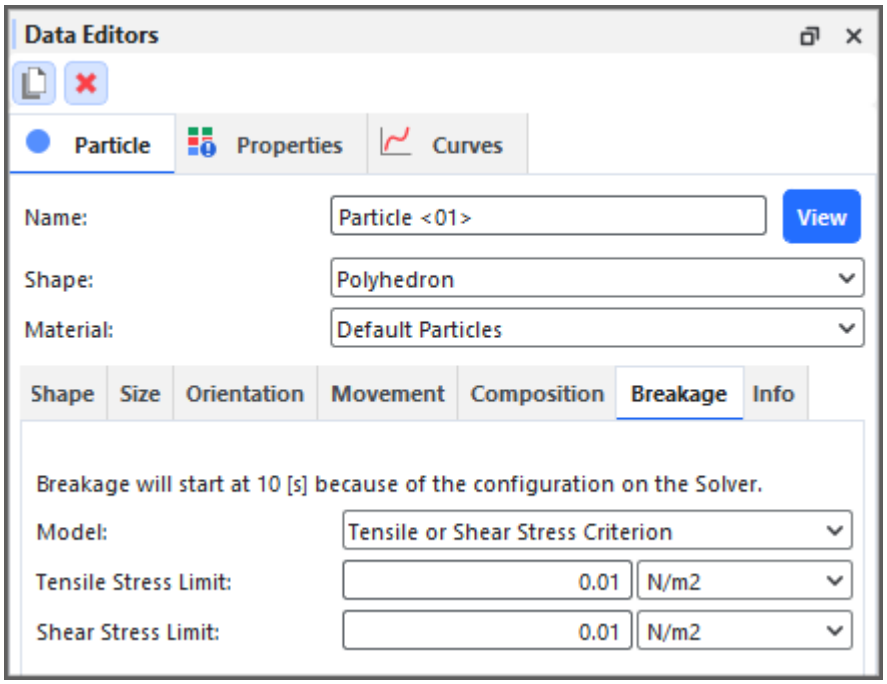
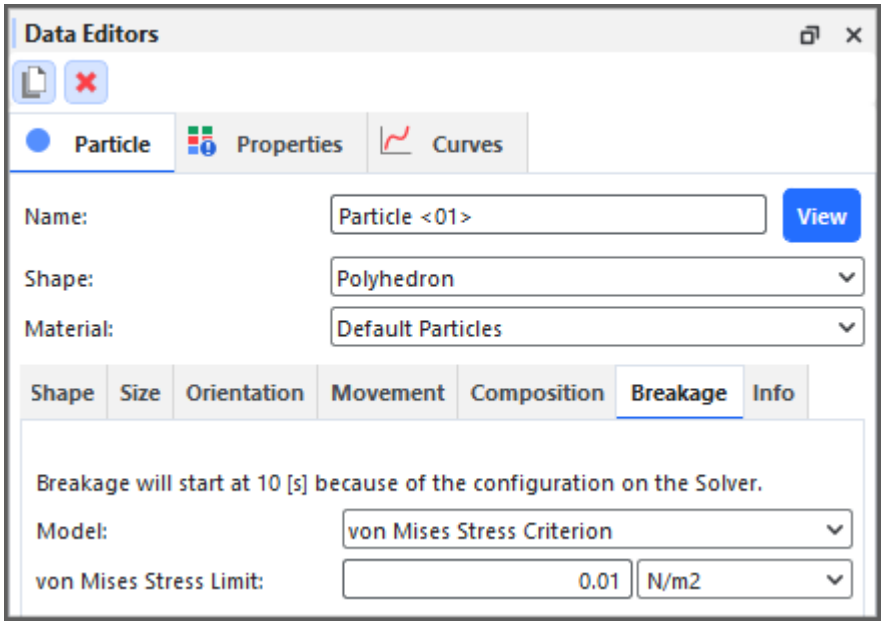


Figure 3.124: Particle, Breakage tab on the Data Editors panel showing the Tensile or Shear Stress Criterion model settings



Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

Figure 3.125: Particle, Breakage tab on the Data Editors panel showing the von Mises Stress Criterion model settings



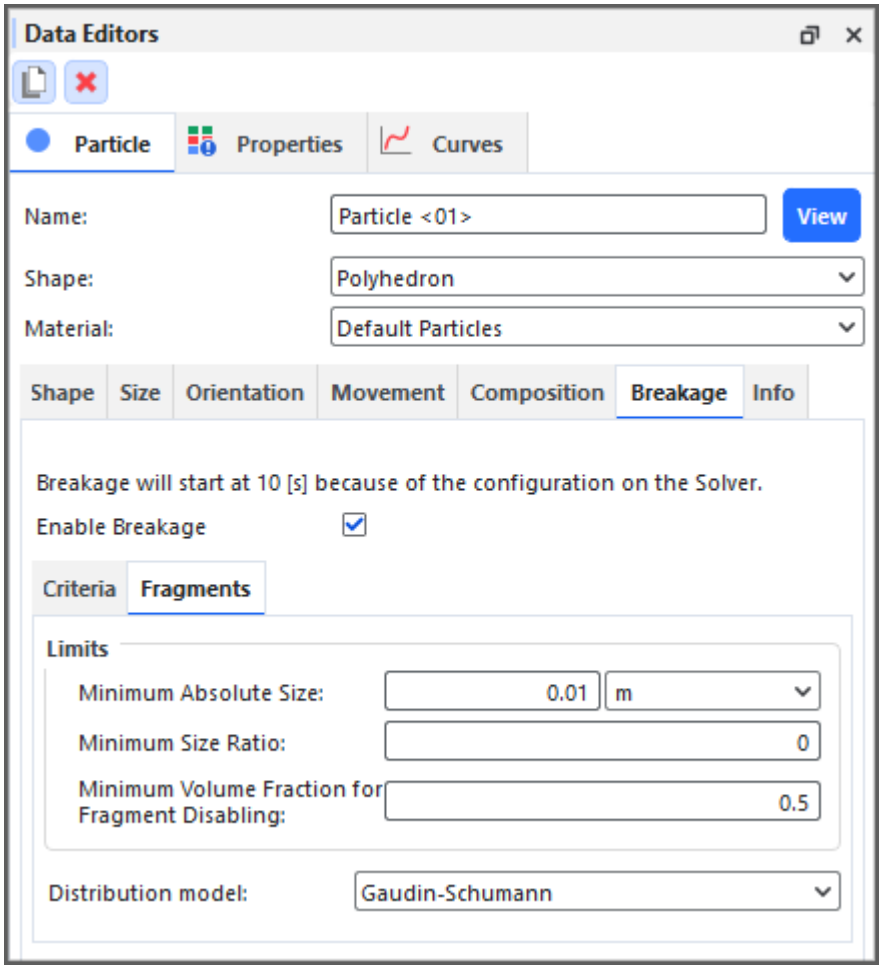
In addition, the Instantaneous Breakage models will have additional options on the **Fragments** sub-tab (Figures 29-30).

Note:

There are no fragment distribution options provided for the Discrete Breakage models.

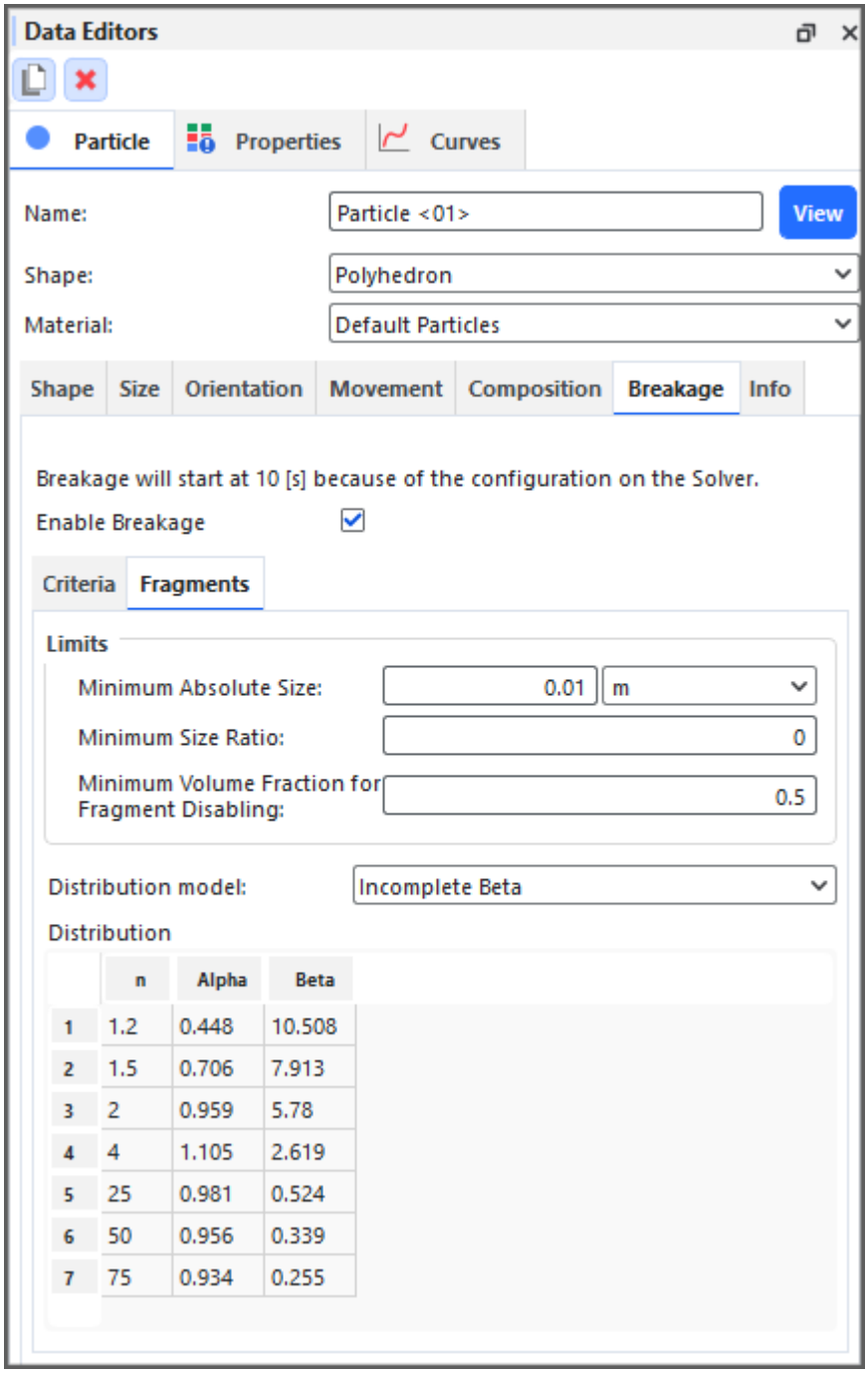
Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

Figure 3.126: Particle, Breakage, Fragments sub-tab on the Data Editors panel showing the Gaudin-Schumann distribution model



Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

Figure 3.127: Particle, Breakage, Fragments sub-tab on the Data Editors panel showing the Incomplete Beta distribution model

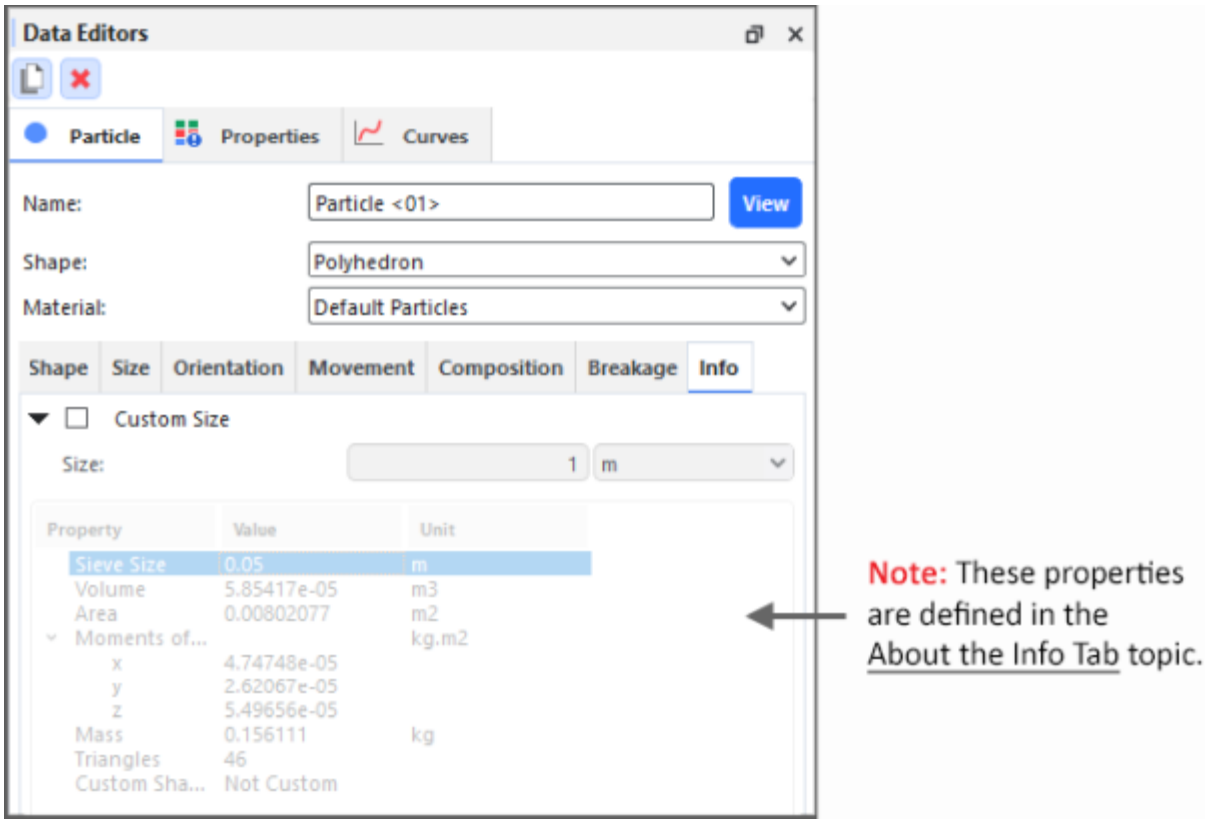


INFO SUB-TAB

The **Info** tab provides more details about the parameter set that you defined, and can help you determine whether the settings you have will meet your needs (Figure 31).

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

Figure 3.128: Particles, Info tab on the Data Editors panel



To keep you from having to switch from tab to tab, a **Custom Size** parameter is provided which when selected, enables you to experiment with alternate particle sizes and immediately see how it affects the property values listed, such as particle **Mass** and number of **Elements**. Once you arrive at a Custom Size you like, you can then go back to the **Size** tab and make your final changes accordingly.

Tips:

- To learn more about the properties listed at the bottom of the Info tab, refer also to the About the Info Tab (p. 83) topic.
- Unlike other Info tabs in Rocky, you can change what is used in the **Unit** column by clicking the unit you want to change and selecting a new option from the list that appears.

DEFORMATION TAB (BETA)

Important:

Note that beta features have not been fully tested and validated. Ansys, Inc. makes no commitment to resolve defects reported against these prototype features. However, your feedback will help us improve the overall quality of the product. We will not guarantee

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

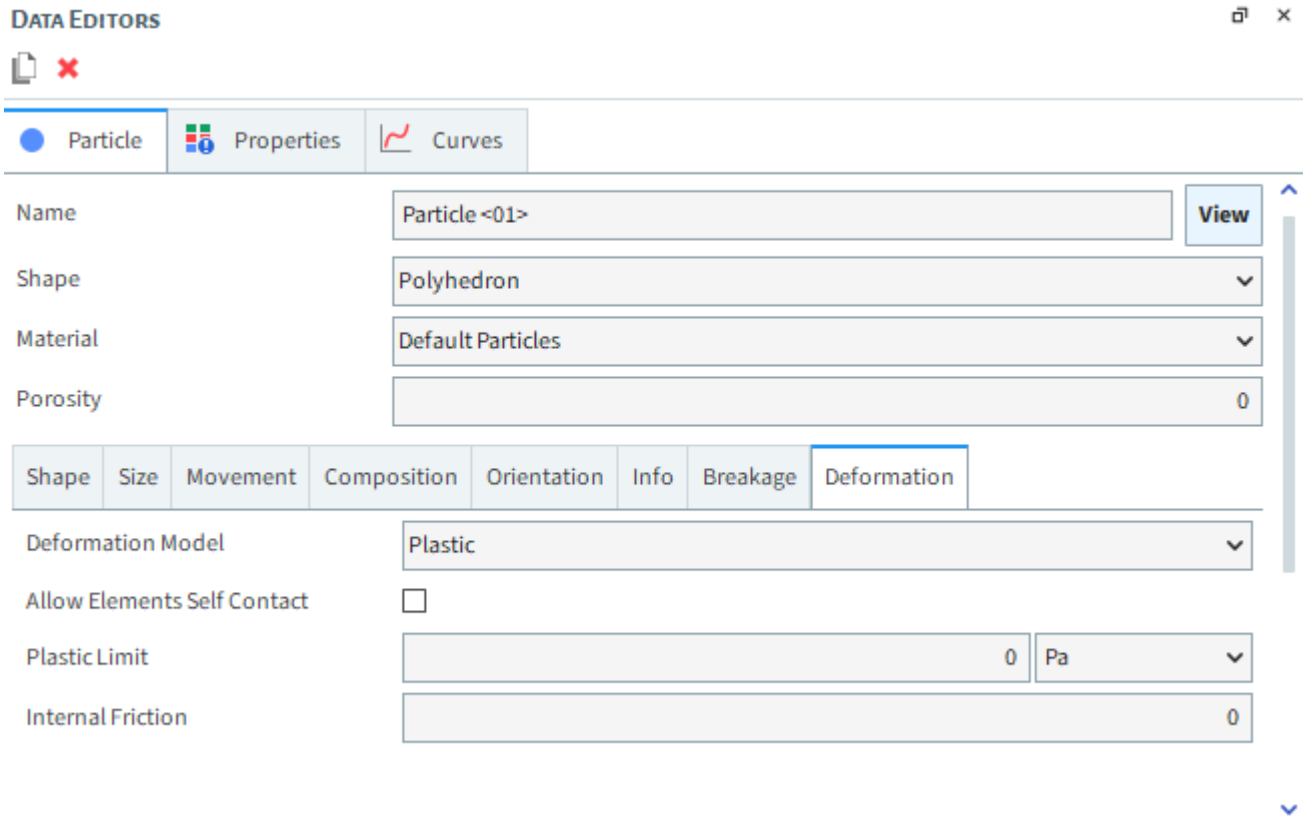
that the projects using this beta feature will run successfully when the feature is finally released so you may, therefore, need to modify the projects.

Note:

The Deformation can only be used if the **Experimental (Beta) Features** checkbox is enabled on the **Options | Preferences | Additional Features** dialog

The **Deformation** tab is a Beta feature for applications involving significant plastic deformation of particles since enables particle large deformation through an FEM-inspired element deformation approach.

Figure 3.129: Particles, Deformation tab (Plastic) on the Data Editors panel



Note:

To consider deformation for the particle's elements in the **Composition** tab select *Multiple Elements*.

In Deformation Model you can select **Elastic** or **Plastic** model. For plastic model, it is necessary to define:

- **Plastic Limit:** which defines a stress limit to account for plasticity.
- **Internal Friction:** which is defined according to the material (=0 for perfectly plastic material).

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

- **Self-contact:** must be enabled if contacts between elements in the same particle are desired.

Important:

In the **Study** entity the **Meshed Particles Upscaling** must be disabled for correct visualization.

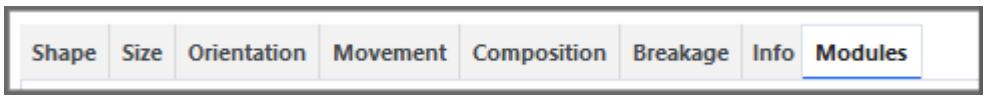
Note:

The Deformation model is available only for solid particle types, including custom polyhedrons.

MODULES SUB-TAB

If you have enabled a module (p. 1392) that affects your individual Particle set settings, you might also have a separate **Modules** sub-tab with additional settings that you can define (Figure 32). Refer to the Module's documentation (if provided) for more information. (See also Rocky Simulation Entities that can be Affected by Modules (p. 223).)

Figure 3.130: Particles, Modules tab on the Data Editors panel



Particle Settings Definitions

See the tables below (and refer to the images above) to help you understand how to add and edit particle sets.

Table 6 : Main Particle tab options (Figure 1 above)

Setting	Description	Range
Name	Enables you to specify a unique identifier for the particular Particle set.	99 character limit
Shape	The initial default (and custom imported) shapes you can use as starting point for defining your final particle shape. Most default shapes have **Shape** attributes that you can use to modify the shape further. (See also About Defining and Importing Custom Particle Shapes (p. 670)).	*Solid* (p. 1395) shapes: • Sphere • Polyhedron • Sphero-Cylinder • Sphero-Polygon • Sphero-Polyhedron • Briquette • Faceted Cylinder • Custom Polyhedron

Setting	Description	Range
		Fiber (p. 1391) shapes: • Straight Fiber • Custom Fiber *Shell* (p. 1395) shapes: • Custom Shell *Assembly* (p. 1389) shapes: • Assembly
Material	Defines the density and Young's Modulus of the Particle set based upon the options you've defined in the Materials list. (See About Modifying Material Compositions (p. 591) for more information.)	List is based upon the Materials that have been defined
Porosity	Defines the particle porosity percentage, according to the particle type.	Positive values

Table 7 : Particle, Shape sub-tab options (Figures 2-6 above)

Setting	Description	Range
Solid and **Fiber** Shape Settings		
Vertical Aspect Ratio	Changes the height (Y value) of the particle assuming a Z value of 1.	Value limited by list box. **Note:** The upper limit for the shapes that allow this setting are provided below: • Polyhedron: 3.75

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

Setting	Description	Range
		• Sphero-Cylinder: 200.05 • Sphero-Polygon: 2.55 • Sphero-Polyhedron: 2.7 • Briquete: 0.6 • Faceted Cylinder: 3.75 • Straight Fiber: 5000.05
Horizontal Aspect Ratio	Changes the width (X value) of the particle assuming a Z value of 1.	Value limited by list box
Smoothness	Changes how smooth or sharp the particle edges are. The higher the value, the smoother the edges.	Value limited by list box
Side Angle	For **Briquettes**, changes the angle of the corners.	Value limited by list box
Number of Corners	Changes how many points a particle's surface has.	Value limited by list box
Superquadric Degree	Changes how square or elliptical a particle appears. The higher the value, the more square the shape. **Note:** The discretization of the particle shape depends on the Aspect Ratio, Superquadric Degree and Number of Corners parameters. The superquadric equation is the same as the ellipsoid equation, but the degree can be higher than 2. The number of corners is placed on the superquadric surface and a convex hull is generated for these corners.	Value limited by list box. **Note:** The upper limit for the shapes that allow this setting are provided below: • Polyhedron: 12.

Setting	Description	Range
		• Sphero-Polyhedron: 12.
Shell Shape Settings		
Thickness	For *Shell* (p. 1395) particles, this determines the current absolute thickness of each 2D plane that makes up the shape, using as its baseline the original size of the imported shape at 100% scale. The value entered is reflected in the **Particles Details** window and affects the resulting mass of the particle as well as the number of *Element* (p. 1390) that are required to make it flexible (also known as *Meshed* (p. 1392)). Important: This value is based only on the original, imported size of the particle and does not take into account changes to the size of your particle. However, a corresponding thickness will be applied to all particles in your set, regardless of their size. **Tip:** For best results, adjust the Thickness only when viewing the particle at 100% **Original Size Scale**. (See Table 6 below.)	Positive values
Assembly Shape Settings		
Particle	For **Assembly** shapes only, this is the *Particle set* (p. 1393) whose shape you want to use as a *part* (p. 1393) within your final Assembly. **Note:** For Particle Assembly the porosity parameter will be defined for each particle that makes up the assembly. Moreover, it is possible to define the porosity in the field **Custom Properties**.	Any Particle set defined as single element (rigid), non-breakable or Straight shape.
PX	For **Assembly** shapes only, this is the part's position on the X axis. Note: The **Particles Details** window shows only the part's position relative to the gravity center of the whole Assembly.	Any value

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

Setting	Description	Range
PY	For **Assembly** shapes only, this is the part's position on the Y axis. Note: The **Particles Details** window shows only the part's position relative to the gravity center of the whole Assembly.	Any value
PZ	For **Assembly** shapes only, this is the part's position on the Z axis. Note: The **Particles Details** window shows only the part's position relative to the gravity center of the whole Assembly.	Any value
Scale	For **Assembly** shapes only, this determines the part's size relative to the largest size of the PSD that is defined on the **Size** tab for the original Particle set. For example: • A value of 1 includes a part equal to the largest size specified for the Particle set on its **Size** tab. • A value of 2 includes a part that is double the size of the largest size specified for the Particle set on its **Size** tab. • A value of 0.5 includes a part that is half the size of the largest size specified for the Particle set on its **Size** tab.	Any value
RX	For **Assembly** shapes only, this is the X component that defines the vector about which the part is rotated. This is used in conjunction with the **Angle** parameter to define the final orientation of the part.	No limit but the values entered will be normalized
RY	For **Assembly** shapes only, this is the Y component that defines the vector about which the part is rotated. This is used in conjunction with the **Angle** parameter to define the final orientation of the part.	No limit but the values entered will be normalized
RZ	For **Assembly** shapes only, this is the Z component that defines the vector about which the part is rotated. This is used in conjunction with the **Angle** parameter to define the final orientation of the part.	No limit but the values entered will be normalized
Angle	For **Assembly** shapes only, this is the degree of rotation about the defined rotation vector (**RX**, **RY**, and **RZ**).	No limit

Table 8: Particle, Size sub-tab options (Figures 7-9 above)

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

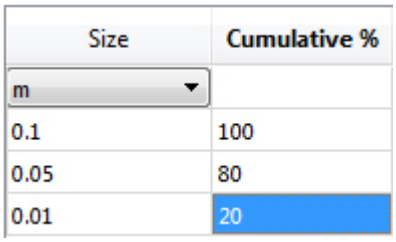
Setting	Description	Range
CGM Scale Factor	When Coarse Grain Modeling is enabled (see also About Physics Parameters (p. 417)), this scale factor will be applied to the particle **Size** (**Diameter** or **Scale Factor**) you enter to determine the final simulated size of the particle and how many smaller particles' behavior it represents. Specifically: • A value of **1.0** will keep the final simulated Size (Diameter or Scale Factor) values the exact same as inputted. For example, entering a **Size** of `1.0 mm` and a **CGM Scale Factor** of `1.0` will simulate particles 1.0 mm in size, each of which represents the behavior of a single particle. • A value **greater than 1.0** will increase the final simulated size of the particle by the scale factor you enter, with each larger particle representing an amount of smaller particles equal to the factor entered. For example, entering a **Size** of `1.0 mm` and a **CGM Scale Factor** of `10` will simulate particles 10.0 mm in size that each represent ten, 1.0 mm particles. **Note:** If more one **Size** row is specified for the Particle set, the same scale factor will be applied to all particle sizes in the set. (See also I cannot find Particle Size Scale Factor (p. 1330)).	1.0 or greater
Size Type	Determines how the size of the particle will be measured. Specifically: • **Sieve Size**: Particle size will be based upon the dimensions of a square hole just big enough for the particle to pass through. This use of virtual "mesh" determines the size of the particles no matter what shape they are. Best used for even and balanced particle shapes such as the **Solid** ones that come by default with Rocky. Should and not be used for fibers or shapes with high aspect ratios. (See also the DEM Technical Manual. (From the Rocky **Help** menu, point to **Manuals**, and then click **DEM Technical Manual**.)) • **Equivalent Sphere Diameter**: Particle size will be based upon the diameter of a sphere of equivalent volume as the shape being measured. Best used for irregular shaped particles. • **Original Size Scale**: For custom (imported) particle shapes, size will be based upon the scale that was imported with the original particle. For default particle shapes, size will be based upon a standard 1.0 m sieve size.	Sieve Size; Equivalent Sphere Diameter Original Size Scale
Size	When **Sieve Size** is set for **Size Type**, this is the dimension of the mesh sieve that determines particle size. Note: For particles composed of only a Single *Element* (p. 1390) (see also *Composition Tab*	Positive values; sizes

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

<table>
<tr><th>Setting</th><th>Description</th><th>Range</th></tr>
<tr><td></td><td>section above), more than one size row can be specified if desired. For particles composed of Multiple Elements, only one size may be specified. If only one size is specified for a particle set, only particles of that size will be used during the simulation. This is because the smallest Size value specified will always have no range.

If more than one size is specified for a particle set, a range of sizes within the Size parameter will be used for each Cumulative % provided. The one exception is for the smallest Size value, which will have no range.

For example, for the sizes and percentages specified below:

 
<table>
<tr><th>Size</th><th>Cumulative %</th></tr>
<tr><td>m</td><td></td></tr>
<tr><td>0.1</td><td>100</td></tr>
<tr><td>0.05</td><td>80</td></tr>
<tr><td>0.01</td><td>20</td></tr>
</table>

• 20% (100-80) of the particles will be in various sizes between 0.05 and 0.1 meters.

• 60% (80-20) of the particles will be in various sizes between 0.01 and.0.05 meters.

• The rest of the particles (20%) will be exactly 0.01 meters in size (no range).

See also Set Particle Size Ranges (p. 375).</td><td>should decrease as you go down the Size table</td></tr>
<tr><td>Diameter</td><td>When Equivalent Sphere Diameter is set for Size Type, this is the diameter of a sphere that has a volume equivalent to that of the particle, which determines particle size. Note: For particles composed of only a Single Element (p. 1390) (see also Composition Tab section above), more than one diameter row can be specified if desired. For particles composed of Multiple Elements, only one diameter row may be specified.

If only one diameter is specified for a particle set, only particles of that equivalent diameter will be used during the simulation. This is because the smallest Diameter value specified will always have no range.

If more than one diameter is specified for a particle set, a range of diameters within the Diameter parameter will be used for each Cumulative % provided. The one exception is for the smallest Diameter value, which will have no range.

For example, for the diameters and percentages specified below:</td><td>Positive values; diameters should decrease as you go down the Diameter table</td></tr>
</table>

<table>
<tr><th>Setting</th><th>Description</th><th>Range</th></tr>
<tr><td></td><td>
<table>
<tr><th>Diameter</th><th>Cumulative %</th></tr>
<tr><td>m</td><td></td></tr>
<tr><td>0.1</td><td>100</td></tr>
<tr><td>0.05</td><td>80</td></tr>
<tr><td>0.01</td><td>20</td></tr>
</table>

20% (100-80) of the particles will be in various sizes equivalent to spheres with diameters between 0.05 and 0.1 meters.
60% (80-20) of the particles will be in various sizes equivalent to spheres with diameters between 0.01 and.0.05 meters.
The rest of the particles (20%) will be sized exactly equivalent to a sphere 0.01 meters in diameter (no range).

See also Set Particle Size Ranges (p. 375).
</td><td></td></tr>
<tr><td>Scale Factor</td><td>When Original Size Scale is set for Size Type, this is the percentage or factor of the original particle size to which the new particle size should be scaled. Note: For particles composed of only a Single Element (p. 1390) (see also Composition Tab section above), more than one scale factor row can be specified if desired. For particles composed of Multiple Elements, only one scale factor row may be specified.

If only one scale factor is specified for a particle set, only particles of that resulting size will be used during the simulation. This is because the smallest Scale Factor value specified will always have no range.

If more than one scale factor is specified for a particle set, a range of resulting sizes within the Scale Factor values will be used for each Cumulative % provided. The one exception is for the smallest Scale Factor value, which will have no range.

For example, for the diameters and percentages specified below:
<table>
<tr><th>Scale Factor</th><th>Cumulative %</th></tr>
<tr><td>%</td><td></td></tr>
<tr><td>100</td><td>100</td></tr>
<tr><td>50</td><td>80</td></tr>
<tr><td>10</td><td>20</td></tr>
</table>

20% (100-80) of the particles will be in various sizes between 50% and 100% of the original size.
60% (80-20) of the particles will be in various sizes between 10% and 50% of the original size.

</td><td>Positive values; factors should decrease as you go down the Scale Factor table</td></tr>
</table>

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

Setting	Description	Range
	• The rest of the particles (20%) will be exactly 10% of the original size. See also Set Particle Size Ranges (p. 375).	
Cumulative %	When more than one size, diameter, or scale factor are specified for a particle set, this is the percentage of particle mass that will be devoted to that particular sieve size, sphere diameter, or original scale factor range. The first row (top most value) should represent the largest particle size in the set. Subsequent rows should represent decreasingly smaller sizes as needed. (See also **Size**, **Diameter**, or **Scale Factor** descriptions above.) See also Set Particle Size Ranges (p. 375).	0<100; values should decrease as you go down the respective **Size**, **Diameter**, or **Scale Factor** tables

Table 9: Particle, Orientation sub-tab options

Setting	Description	Range
Orientation		
Method	Enables you to select how you want to define the orientation of the particle shape. Specifically: • **Angles** enables you to define angles of rotation in three directions, the order of which you can also specify. • **Angle and Vector** enables you define one vector and one angle of rotation around it. • **Basis Vectors** enables you to define the X, Y, and Z directions of the particles's local basis.	Angles; Angle and Vector; Basis Vectors.
Method Angles		
Order	When **Angles** is selected for **Orientation**, this defines the order in which the three **Rotation** text fields will be applied.	XYZ; ZXY; YXZ; YZX; ZXY; ZYX.

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

Setting	Description	Range
Local Angles	When **Angles** is selected for **Orientation**, this defines what coordinate system will be used as a basis for the angle specified. Specifically: • When enabled, the angle will be based on the local coordinate system. • When cleared, the angle will be based on the global coordinate system.	Turns on or off
Rotation	When **Angles** is selected for **Orientation**, this is the degree of particle rotation in each of the three directions specified by the **Order** provided.	Any value
MethodAngle and Vector		
Angle	When **Angle and Vector** is selected for **Orientation**, this is the angle the cube will rotate around the **Vector** defined.	Any value
Vector	When **Angle and Vector** is selected for **Orientation**, this is the X, Y, and Z components that define the vector around which the particles will rotate, using the **Angle** defined.	No limit but the values entered will be normalized
MethodBasis Vectors		
X direction	When **Basis Vectors** is selected for **Orientation**, this is the coordinate values defining the first of three directional vectors that together define the final orientation of the particle. **Tip:** To ensure correct results, make sure you define this vector as orthogonal (perpendicular) to the other two vectors.	No limit but the values entered will be normalized
Y direction	When **Basis Vectors** is selected for **Orientation**, this is the coordinate values defining the second of three directional vectors that together define the final orientation of the particle. **Tip:** To ensure correct results, make sure you define this vector as orthogonal (perpendicular) to the other two vectors.	No limit but the values entered will be normalized

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

Setting	Description	Range
Z direction	When **Basis Vectors** is selected for **Orientation**, this is the coordinate values defining the third of three directional vectors that together define the final orientation of the particle. **Tip:** To ensure correct results, make sure you define this vector as orthogonal (perpendicular) to the other two vectors.	No limit but the values entered will be normalized
Random Angle		
Half Range (x, y, z)	Defines the X, Y, and Z angle limits within which Rocky will randomly orient each individual particle within the Particle set. Tips: • For a completely random orientation in all directions, set your values to 180, 180, 180. • Values above 180 are allowed but will result in the initial angles having a larger probability of selection than the later angles.	No limit

Table 10: Particle, Movement sub-tab options (Figure 15-15a above)

Setting	Description	Range
Rolling Resistance	When a **Rolling Resistance Model** is defined on the **Physics | Momentum** tab (see also About Physics Parameters (p. 417)), this setting changes how particle shapes that are both rigid and convex roll on the boundary surface. A higher value reduces rolling. Tips: • Works best for Sphere particle shapes. • Does not work well for the following shape types and situations and is therefore best to set to zero (0) in these cases: – When you are defining an elongated particle shape (i.e., shapes with high aspect ratios). – When you are defining a *shaped particle* (p. 1395) in a project that has the Tangential Force model set to **Mindlin- Deresiewicz** as doing so can lead to an	0 - 1

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

Setting	Description	Range
	unstable Angle of Repose. (See also Physics and Force Limitations (p. 55)). **Note:** Neither Flexible particles nor concave-shaped rigid particles support editing this value; as a result, it will always be set to zero (0) for these shapes. See Also: • About Physics Parameters (p. 417) • Change Particle Adhesion (p. 374)	
Enable Rotations	When selected for rigid particle shapes, allows the particle to rotate during the simulation. **Note:** Flexible particles do not support editing this value; as a result, it will always be selected (enabled) for these shapes.	Turns on or off
Change Mass Properties	When enabled (for non-Assembly shapes only), this allows you to change the position of the particle's center of mass and moment of inertia values. Tips: • You only need to define these values for particles whose center of mass does not coincide with its geometric center, such as in cases where the density of the particle is not homogeneous or the particle is composed by more than one material. • If the particle's center of mass is located at its geometric center and your particle's mass is uniformly distributed, you do not need to enable this checkbox. Rocky will automatically compute the center of mass and the moment of inertia based on the particle's geometry and density.	Turns on or off
Center of Mass Offset	When **Change Mass Properties** is enabled, these three values specify the displacement of the particle's center of mass relative to its geometric center in the X, Y, and Z directions, expressed as percentages of the particle's sieve size. This relationship can be described as:	-50% to +50%

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

Setting	Description	Range
	$\delta_{CM}=(\phi_x,\phi_y,\phi_z)L$ Where: δ_{CM} is the required displacement of the center of mass relative to the particle's geometric center, ϕ_x,ϕ_y,ϕ_z are the offset values in the directions X, Y, Z, respectively, and L is the particle's sieve size. For example, if a particle of **Sieve Size** `1m` needs to have its center of mass displaced to `(0,-0.2,0)m`, the **Center of Mass Offset** values you enter here should be `(0,-20,0)`%.	
Principal Moment of Inertia	When **Change Mass Properties** is enabled, these are the particle's principal moments of inertia along the X, Y, and Z axes defined below. The values you enter should be defined based upon the **Center of Mass Offset** position you have set, and should assume the particle has a sieve size equal to `1m`. **Note:** The units here indicate that the values you provide should be the moment of inertia divided by the density. This is different than the **Moment of Inertia** Property calculated during post-processing. (See also About Properties (p. 933)).	Positive values greater than but not equal to zero
X direction	The X, Y, and Z components that define the X axis for the **Principal Moment of Inertia**. Note: This is based upon the global coordinate system.	No limit but the values entered will be normalized, and the base must be positive orthonormal
Y direction	The X, Y, and Z components that define the Y axis for the **Principal Moment of Inertia**. Note: This is based upon the global coordinate system.	No limit but the values entered will be normalized, and the base must be positive orthonormal
Z direction	The X, Y, and Z components that define the Z axis for the **Principal Moment of Inertia**. Note: This is based upon the global coordinate system.	No limit but the values entered will be normalized, and the base must be positive orthonormal

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

Table 10a: Particle, Custom Properties sub-tab options (Figure 15b above)

Setting	Description	Range
Change Assembly Properties	When enabled for Assembly particle shapes, this allows you to enter the values of Area, Volume, Geometric Center, Center of Mass and other properties from the CAD program with which you designed your Assembly particle shape.	Turns on or off
Area	The Area of your Assembly particle.	Positive values greater than but not equal to zero.
Volume	The Volume of your Assembly particle.	Positive values greater than but not equal to zero.
Geometric Center	The Geometric Center you want to set for your particle assembly.	Coordinates inside the particle bounding box.
Center of Mass	The Center of Mass you want to set for your particle assembly.	Coordinates inside the particle bounding box.
Moment of Inertia	When **Change Assembly Properties** is enabled, these are the particle's principal moments of inertia along the X, Y, and Z axes defined below. The values you enter should be defined based upon the **Center of Mass** position you have set, and should assume the particle has an Original Size Scale equal to `1m`. **Note:** The units here indicate that the values you provide should be the moment of inertia divided by the density. This is different than the **Moment of Inertia** Property calculated during post-processing. (See also About Properties (p. 933)).	Positive values greater than but not equal to zero
X direction	The X, Y, and Z components that define the X axis for the **Principal Moment of Inertia**. Note: This is based upon the global coordinate system.	No limit but the values entered will be normalized, and the base must be positive orthonormal
Y direction	The X, Y, and Z components that define the Y axis for the **Principal Moment of Inertia**. Note: This is based upon the global coordinate system.	No limit but the values entered will be normalized, and the base must be positive orthonormal
Z direction	The X, Y, and Z components that define the Z axis for the **Principal Moment of Inertia**. Note:	No limit but the values entered will be normalized, and the base

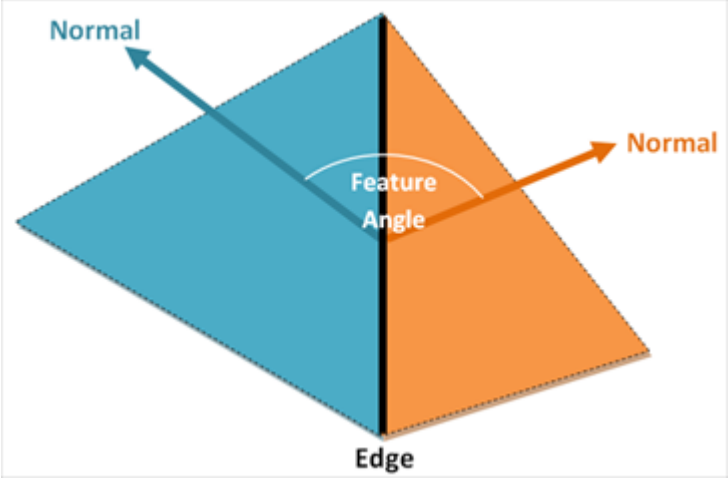
Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

Setting	Description	Range
	This is based upon the global coordinate system.	must be positive orthonormal

Table 11: Particle, Composition sub-tab options (Figure 16-20 above)

Setting	Description	Range
Composition	Determines whether the particle will be composed of one or many *Element* (p. 1390). Specifically: • A **Single Element** particle is composed as an individual, solid shape. In this version of Rocky, these kinds of particle compositions cannot be made flexible. • A **Multiple Elements** particle is *Meshed* (p. 1392) to comprise it of many smaller *Element* (p. 1390) "glued" together. Composing the particle in this way provides several unique benefits, including flexibility, and/or a more detailed particle analysis. **Note:** Only Fibers, Shells, and Polyhedron and Custom Polyhedron (Solid) shapes are currently able to be meshed. (See also Program Limitations (p. 52)).	Single Element; Multiple Elements
Remesh to Target	For *Shell* (p. 1395) particles only, defines how the triangular elements that make up the shape surface are determined. Specifically: • When enabled, the shape will be *remeshed* (p. 1394) to match as closely as possible the **Target Number of Elements** value. • When cleared, the shape will retain the original meshing of the imported .stl file.	Turns on or off
Target Number of Elements	When **Multiple Elements** are selected, this allows you to (roughly) control the number of individual *Element* (p. 1390) that will comprise your particle shape. The higher the value, the more accurate the representation but the longer the processing time due to the increased calculations. Tips: • Learn whether you have entered less than the minimum amount of elements required by checking for Particle errors at the bottom of the Data Editors panel or on the Status panel (Figure 13). • You can discover the exact number of Elements Rocky plans to use by viewing the **Elements** value on the **Info** sub-tab. • For a Custom *Fiber* (p. 1391), the minimum amount of Elements is equal to the number of *Segment* (p. 1394) that were defined. Setting	Positive values

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

Setting	Description	Range
	a value smaller than the number of Segments will result in a shape that when flexible, bends only at the intersections of the nodes making up the Segments.	
Feature Angle	For *Solid* (p. 1395) and and *Shell* (p. 1395) particles, when **Multiple Elements** are selected, this sets the angle limit at which the meshing algorithm in Rocky will try to preserve the edges of the triangular components that make up the base shape when turning them into individual *Element* (p. 1390). An edge is defined as the line that connects two triangles and the feature angle is the angle between the two triangles' normals (Figure 21).  **Figure 21: Feature Angle diagram** Specifically: • Setting zero (0) (sides are parallel) preserves every original edge. Note: This may require increasing the **Target Number of Elements** amount. • Setting 180 (sides coincide) ignores every original edge. • Setting a value between zero (0) and 180 will preserve some edges and ignore others, depending upon the shape. Changing this setting is useful for particle shapes containing triangles with sharp edges where the default meshing calculations generate individual Element shapes that change the sharp parts of the original particle shape. in an undesirable way (Figure 14). **Tip:** You can visualize the shape of the individual Elements in the Particles Details window by turning on **Edges** from the Coloring tab.	0-180

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

Setting	Description	Range
Elasticity	When **Multiple Elements** are selected, this determines how the elasticity of the *joint* (p. 1392) between the individual *Element* (p. 1390) are treated. Specifically: • When **Isotropic** is selected, the joints will have the same Young's modulus (defining the same **Elastic Ratio**) in all directions of deformation. • When **Anisotropic** is selected, you can then define different **Elastic Ratio** values in different directions for the joints, which in so doing, allows a different Young's modulus to be considered in each direction.	Isotropic; Anisotropic
Elastic Ratio	When **Multiple Elements** are selected, and **Isotropic** is selected for **Elasticity**, this determines the degree of flexibility of the particle. Formally, this parameter is defined as the ratio of the Young's modulus of the joints to the bulk Young's modulus defined for the particle's material. (See also About Modifying Material Compositions (p. 591)). **Tip:** For most cases, to avoid either too soft or too rigid particles, values close to 1 are optimal.	10e-5 - 100
Elastic Ratio Normal	When **Anisotropic** is selected for **Elasticity**, this defines the degree of flexibility in the Normal direction.	10e-5 - 100
Elastic Ratio Tangential	When **Anisotropic** is selected for **Elasticity**, this defines the degree of flexibility in the Tangential direction.	10e-5 - 100
Elastic Ratio Bending	When **Anisotropic** is selected for **Elasticity**, this defines the degree of flexibility when the particle is bent.	10e-5 - 100
Elastic Ratio Torsion	When **Anisotropic** is selected for **Elasticity**, this defines the degree of flexibility when the particle is twisted.	10e-5 - 100
Damping Ratio Joint	When **Multiple Elements** are selected, this determines the degree of decay of oscillations of the *joint* (p. 1392) between the individual Elements that compose the particle, and is applied as joint forces and moments proportional to the respective joint deformation velocities. It is effective in reducing or removing high-frequency oscillations.	0 - 1
Damping Ratio Element	When **Multiple Elements** are selected, this determines the degree of decay of oscillations among the Elements that compose the particle, and is applied as Element forces proportional to difference between the Element velocity and the velocity of the center of mass of the	0 - 1

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

Setting	Description	Range
	particle to which the Element belongs. It is effective in reducing or removing oscillations of low frequency.	
Conductivity Ratio	When both the **Thermal Model** is enabled (see also About Physics Parameters (p. 417)) and **Multiple Elements** are selected, this enables you to increase or reduce the heat transfer between *Element* (p. 1390) relative to the heat transfer associated to collisions with other particles or boundaries.	0.01 - 100
Joint Model	When **Multiple Elements** are selected, this defines the plasticity model being used to simulate the movements between the *joint* (p. 1392) of the individual *Element* (p. 1390). Specifically: • **Linear Elastic** can apply to all types of flexible particles. In this model, when a joint experiences linear or angular deformations, it reacts with forces or moments directly proportional to those deformations. • **Bilinear Elastoplastic** can apply only to flexible *Fiber* (p. 1391) and *Shell* (p. 1395), and affects only angular deformations (bending and torsion). After reaching a limiting value of the angular deformation (which you can define via the **Angle Limit** parameters provided), the joint response is elastoplastic, which means that a portion of the deformation will be permanent. **Note:** If you have an *external Module* that adds in an additional Joint Model, you must use that model at least once in your simulation project if the Module is enabled. Refer to the Module's documentation (if provided) for more information. (See also Rocky Simulation Entities that can be Affected by Modules (p. 223)).	Linear Elastic; Bilinear Elastoplastic
Plastic Ratio	When **Multiple Elements** is selected and also **Bilinear Elastoplastic** is selected for **Joint Model**, this defines the ratio between the stiffness in the elastoplastic regime to the stiffness in the elastic regime. The smaller the value of this ratio, the more evident will be the permanent deformations on the joints.	0 - 1
Torsion Angle Limit	For *Shell* (p. 1395) and *Fiber* (p. 1391) particle shapes only, when **Multiple Elements** is selected and also **Bilinear Elastoplastic** is selected for **Joint Model**, this defines the limit of the purely elastic regime on the bilinear elastoplastic model, for torsion deformation.	0 - 360 degrees
Bending Angle Limit	For straight and custom *Fiber* (p. 1391) particle shapes only, when **Multiple Elements** is selected and also **Bilinear Elastoplastic** is selected for **Joint Model**, this defines the limit of the purely elastic regime on the bilinear elastoplastic model, for bending deformations.	Positive values
1st Bending	For custom *Shell* (p. 1395) particle shapes only, when **Multiple Elements** is selected and also **Bilinear Elastoplastic** is selected for **Joint Model**, this defines the limit of the purely elastic regime on the bilinear	0 - 180 degrees

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

Setting	Description	Range
Angle Limit	elastoplastic model, for bending deformations in the direction perpendicular to the thickness of the shell *Element* (p. 1390).	
2nd Bending Angle Limit	For custom *Shell* (p. 1395) particle shapes only, when **Multiple Elements** is selected and also **Bilinear Elastoplastic** is selected for **Joint Model**, this defines the limit of the purely elastic regime on the bilinear elastoplastic model, for bending deformations in the direction parallel to the thickness of the shell *Element* (p. 1390).	0 - 180 degrees
With Failure	For custom *Fiber* (p. 1391) and *Shell* (p. 1395) particle shapes only, when **Multiple Elements** is selected and also **Bilinear Elastoplastic** is selected for **Joint Model**, this allows you to add a sudden drop on the joint resistance to deformations when the angular deformation reaches the limiting value that controls the elastoplastic regime transition.	Turns on or off
Failure Ratio	For custom *Fiber* (p. 1391) and *Shell* (p. 1395) particle shapes only, when **Multiple Elements** is selected and also **Bilinear Elastoplastic** is selected for **Joint Model**, this allows you to control the amount of drop on the joint resistance when the angular deformation reaches the limiting value that controls the elastoplastic regime transition.	0 - 1

Table 12: Particle, Breakage sub-tab options for Single Element Particles (Figures 21-23 and 29-31 above)

Setting	Description	Range
Enable Breakage	For Particle sets composed of Single Elements, this determines whether *Instantaneous Breakage* calculations will be taken into account during the simulation.	Turns on or off
Criteria tab	tab	tab
Model	Defines which method of *Instantaneous Breakage* calculations will be enabled during the simulation. • **Ab-T10**: *Instantaneous Breakage* calculations will be applied according to the Ab-T10 model used in all previous versions of Rocky. • **Tavares**: *Instantaneous Breakage* calculations will be applied according to the model developed by Professor Tavares. **Important:** Both Shape and Composition determine whether a particle set supports breakage	Ab-T10; Tavares Note: If you have an *external Module* that adds in an additional Breakage Model, you must use that model at least once in your simulation project if the Module is enabled. Refer to the Module's documentation (if provided) for more information. (See

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

Setting	Description	Range
	and the models available. (See also About Particle Breakage (p. 665)). **Tip:** Refer to DEM Technical Manual for further detail on the specific models. (From the Rocky **Help** menu, point to **Manuals**, and then click **DEM Technical Manual**.)	also Rocky Simulation Entities that can be Affected by Modules (p. 223)).
Ab-T10 Model Reference Size	This is the reference size used in drop weight tests. Typically, this size is 25 to 40 mm. This value and the **Reference Minimum Specific Energy** (for reference size) value are together used to compute the minimum specific energy for the particle.	Any positive value
Ab-T10 Model Reference Minimum Specific Energy	This is the minimum specific energy for the **Reference Size** value. Both values are used to calculate the minimum specific energy for the particle. If the total specific energy for a particle that comes into contact with another particle or geometry is less than this value, breakage will not occur. This value is especially important in cases where particles break primarily on multiple impacts, such as in SAG or AG mills. For an instantaneous impact breakage event, such as with impact crushers or HPGRs, this value is not important and can set to zero.	Any positive value (zero is recommended)
Ab-T10 Model Selection Function Coefficient	The particle breakage strength parameter, which is a material constant. Reducing this value decreases the rate of breakage. Note: This is "S" in the model equation referenced in the Potapov paper but is typically seen as "b" in other Ab-T10 equations.	Any positive value
Ab-T10 Model Maximum t10 Value	Maximum degree of breakage. Though this value does not affect how easily a particle will break, it does affect the size of the resulting particles after breakage. The higher the value, the finer the fragments will be after breakage. Note: This is "M" in the model equation referenced in the Potapov paper but is typically seen as "A" in other Ab-T10 equations.	0-100

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

Setting	Description	Range
Tavares Model σ^2	Refers to the σ^2 (sigma squared) value of the Tavares breakage equation. (See the Rocky **DEM Technical Manual** for further details.)	Positive values
Tavares Model $e_{\max}/e_{50}$	Refers to the Ratio $e_{\max}/e_{50}$ value of the Tavares breakage equation. (See the Rocky **DEM Technical Manual** for further details.)	Positive values
Tavares Model$e_{\min}$ e min	Refers to the minimum specific energy for breakage value used in the Tavares breakage equation. (See the Rocky **DEM Technical Manual** for further details.)	Positive values
Tavares Model e_∞	Refers to the e_∞ (e infinity) value of the Tavares breakage equation. (See the Rocky **DEM Technical Manual** for further details.)	Positive values
Tavares Model d_0d0	Refers to the d_0 value of the Tavares breakage equation. (See the Rocky **DEM Technical Manual** for further details.)	Positive values
Tavares Model φ	Refers to the φ (phi) value of the Tavares breakage equation. (See the Rocky **DEM Technical Manual** for further details.)	Positive values
Tavares Model γ	Refers to the γ (gamma) value of the Tavares breakage equation. (See the Rocky **DEM Technical Manual** for further details.)	Positive values
Tavares Model AA	Refers to the A value of the Tavares breakage equation. (See the Rocky **DEM Technical Manual** for further details.)	Positive values
Tavares Model b'b'	Refers to the b' value of the Tavares breakage equation. (See the Rocky **DEM Technical Manual** for further details.)	Positive values
Tavares Model Fragments Energy Factor	Multiplies the value of the instantaneous specific impact energy by this factor. (See the Rocky **DEM Technical Manual** for further details.)	Any value (Recommended range is between 0-1.)
Tavares Model Energy for t10 Calculation	Defines how the specific fracture energy will be determined when calculating the value of the t10 parameter. (See the Rocky **DEM Technical Manual** for further details.)	Current Particle Energy; Median Specific Fracture Energy (e50b)
Fragments tab		
Limits Minimum Absolute Size	Smallest size the particle fragments can be after breakage is applied.	Positive values

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

Setting	Description	Range
Limits Minimum Size Ratio	Smallest size the particle fragments can be at each breakage incident, relative to the parent particle (either a whole particle or a fragment).	0-1 Note: Values are dimensionless
Limits Minimum Volume Fraction for Fragment Disabling	Defines the minimum volume fraction that a broken particle (fragment) can have before being considered too small to be included in calculations. (See the Rocky **DEM Technical Manual** for further details.)	0-1
Distribution model	Type of fragment distribution size used in the model. Specifically: • **Gaudin-Schumann** can only be used by the Ab-T10 or Tavares breakage models. • **Incomplete Beta** can only be used by the Ab-T10 or Tavares breakage models. (See the Rocky **DEM Technical Manual** for further details.)	Gaudin-Schumann; Incomplete Beta Note: If you have an *external Module* that adds in an additional Distribution Model, you must use that model at least once in your simulation project if the Module is enabled. Refer to the Module's documentation (if provided) for more information. (See also Rocky Simulation Entities that can be Affected by Modules (p. 223)).
Distribution	When **Incomplete Beta** is selected for **Distribution model**, this displays in table rows separate n, Alpha, and Beta value sets, which Rocky then uses with the t_{10} value previously calculated to determine the product size distribution resulting from particle breakage. n is the independent variable chosen for expressing this distribution, with t_n interpreted as the percentage of fragments with size less or equal than $\frac{1}{n}$ of the original size of the particle. For each pair of Alpha and Beta values provided, a value of t_n is calculated according to the Incomplete Beta function; any other intermediate value of t_n falling outside the calculated values are obtained via linear interpolation. Specifically: • **n** is the subscript number n referenced in t_n of the Incomplete Beta function.	Positive values Note: Values are dimensionless

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

Setting	Description	Range
	• **Alpha** is the α_n value of the Incomplete Beta function for the corresponding n. • **Beta** is the β_n value of the Incomplete Beta function for the corresponding n. (See the Rocky **DEM Technical Manual** for further details.) **Tip:** While the table is limited by the seven rows provided, you can edit the values by clicking the value you want to change, and then typing the new value you want.	

Table 13: Particle, Breakage sub-tab options for Multi-Element Particles (Figures 24-28 above)

Setting	Description	Range
Model	Defines whether *Discrete Breakage* calculations will be enabled during the simulation and if so, which method will be used. • **None**: No breakage calculations will be applied during the simulation. • **Griffith Energy Criterion**: Available only for particle sets composed of Multiple Elements (see also the *Composition Tab* section above), *Discrete Breakage* calculations will be applied according to Griffith's criterion. • **Shear Stress Criterion**: Available only for particle sets composed of Multiple Elements (see also the *Composition Tab* section above), *Discrete Breakage* calculations will be based on the value of the shear stress experienced by the *joint* (p. 1392). • **Tensile Stress Criterion**: Available only for particle sets composed of Multiple Elements (see also the *Composition Tab* section above), *Discrete Breakage* calculations will be based on the value of the tensile stress experienced by the *joint* (p. 1392). • **Tensile or Shear Stress Criterion**: Available only for particle sets composed of Multiple Elements (see also the *Composition Tab* section above), *Discrete Breakage* calculations will be based on a	None; Griffith Energy Criterion; Shear Stess Criterion; Tensile Stress Criterion; Tensile or Shear Stress Criterion; von Mises Stress Criterion. Note: If you have an *external Module* that adds in an additional Breakage Model, you must use that model at least once in your simulation project if the Module is enabled. Refer to the Module's documentation (if provided) for more information. (See also Rocky

Setting	Description	Range
	combination of the tensile stress and shear stress criteria explained above. In this option, both criteria are verified at the same time, and when either one of them is satisfied, the *joint* (p. 1392) breaks. • **von Mises Stress Criterion**: Available only for particle sets composed of Multiple Elements (see also the *Composition Tab* section above), *Discrete Breakage* calculations will be based on the value of the equivalent von Mises stress, which is a combination of the tensile and shear stresses. **Important:** Both Shape and Composition determine whether a particle set supports breakage and the models available. (See also About Particle Breakage (p. 665)). **Tip:** Refer to DEM Technical Manual for further detail on the specific models. (From the Rocky **Help** menu, point to **Manuals**, and then click **DEM Technical Manual**.)	Simulation Entities that can be Affected by Modules (p. 223)).
Griffith Energy Criterion Model Surface Energy	Amount of surface energy used by the Griffith breakage model to calculate the energy balance upon which the model is based. The larger the value of the surface energy, the more resistant to breakage the *joint* (p. 1392) will be.	Positive values
Shear Stress Criterion Model Shear Stress Limit	Value that defines when a joint breaks in the Shear Stress breakage model. Whenever the shear stress experienced by a joint is larger than this limit, the joint will break.	Positive values
Tensile Stress Criterion Model Tensile Stress Limit	Value that defines when a joint breaks in the Tensile Stress breakage model. Whenever the tensile stress experienced by a joint is larger than this limit, the joint will break.	Positive values
Tensile or Shear Stress	Value that defines the upper limit for tensile stress that can be experienced by a joint before breaking. In this model, the joint will break whenever either	Positive values

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

Setting	Description	Range
Criterion Model Tensile Stress Limit	this or the shear stress limit is exceeded; whichever limit is reached first will determine breakage.	
Tensile or Shear Stress Criterion Model Shear Stress Limit	Value that defines the upper limit for shear stress that can be experienced by a joint before breaking. In this model, the joint will break whenever either this or the tensile stress limit is exceeded; whichever limit is reached first will determine breakage.	Positive values
von Mises Stress Criterion Model von Mises Stress Limit	Value that defines the upper limit for von Mises stress - which is a combination of the tensile and shear stresses - that can be experienced by a joint before breaking.	Positive values
Include Rotational Deformations	Available only for *Fiber* (p. 1391) and *Shell* (p. 1395) particle sets composed of Multiple Elements (see also the *Composition Tab* section above), this determines whether the stresses caused by rotational deformations (bending or torsion) are included when considering *Discrete Breakage* criteria.	Turns on or off

Table 14: Particle, Info sub-tab options (Figure 32 above)

Setting	Description	Range
Custom Size	When **Sieve Size** is selected for **Size Type** on the **Size** sub-tab, this allows you to experiment with how different particle sizes affect certain Particle properties-such as volume, mass, and number of *Element* (p. 1390) (see also About Properties (p. 933))- without switching back and forth to different Particle tabs. When enabled, Property Values (bottom of Info tab) are calculated according to the **Size** value entered. When cleared, Property Values reflect the information entered on the **Size** sub-tab.	Turns on or off
Size	When **Custom Size** is enabled, the value you enter here re-calculates the Property Values listed at the bottom of the Info tab. **Tip:** When you arrive at a size that meets your needs, ensure you change the real size value on the **Size** sub-tab. Only	No limit

Setting	Description	Range
	the **Size** tab value, and not this experimental value, is used during the simulation. **Note:** This text field does not support parametric variables. (See also I cannot enter an input variable or mathematical function into a text field (p. 1334)).	
Custom Diameter	When **Equivalent Sphere Diameter** is selected for **Size Type** on the **Size** sub-tab, this allows you to experiment with how different particle diameters affect certain Particle properties-such as volume, mass, and number of *Element* (p. 1390) (see also About Properties (p. 933))- without switching back and forth to different Particle tabs. When enabled, Property Values (bottom of Info tab) are calculated according to the **Diameter** value entered. When cleared, Property Values reflect the information entered on the **Size** sub-tab.	Turns on or off
Diameter	When **Custom Diameter** is enabled, the value you enter here re-calculates the Property Values listed at the bottom of the Info tab. **Tip:** When you arrive at a diameter that meets your needs, ensure you change the real diameter value on the **Size** sub-tab. Only the **Size** tab value, and not this experimental value, is used during the simulation.	No limit
Custom Scale Factor	When **Original Scale Size** is selected for **Size Type** on the **Size** sub-tab, this allows you to experiment with how different particle scale factors affect certain Particle properties-such as volume, mass, and number of *Element* (p. 1390) (see also About Properties (p. 933))- without switching back and forth to different Particle tabs. When enabled, Property Values (bottom of Info tab) are calculated according to the **Scale Factor** value entered. When cleared, Property Values reflect the information entered on the **Size** sub-tab.	Turns on or off
Scale Factor	When **Custom Scale Factor** is enabled, the value you enter here re-calculates the Property Values listed at the bottom of the Info tab. **Tip:** When you arrive at a scale factor that meets your needs, ensure you change the real scale factor value on the **Size** sub-tab. Only the **Size** tab value, and not this experimental value, is used during the simulation.	No limit

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

Table 15: Particle, Module sub-tab options (Figure 33 above)

Setting	Description	Range
Modules		
(Varies)	These settings are specific to only certain *external Module* (p. 1391) and are therefore not documented in the Rocky User Manual. Refer to the Module's documentation (if provided) for more information. (See also Rocky Simulation Entities that can be Affected by Modules (p. 223)).	(Varies)

What would you like to do?

- Add a New Particle Set (p. 678)
- Modify the Parameters of a Particle Set (p. 678)
- Preview a Particle Shape in 3D (p. 681)
- Remove an Existing Particle Set (p. 698)
- Export a Particle Shape to an STL File (p. 698)
- Enable and View Collision Statistics for Particle Surfaces (Intra) (p. 346)
- Learn more About Particles Details Windows (p. 679)

See Also:

- Rocky Particle Shapes and Capabilities (p. 41)
- About Defining and Importing Custom Particle Shapes (p. 670)
- Create and View a Particle Assembly (p. 398)
- Set Simulation Parameters (p. 414)
- A fiber or other particle shape disappeared from my simulation during processing (p. 1357)

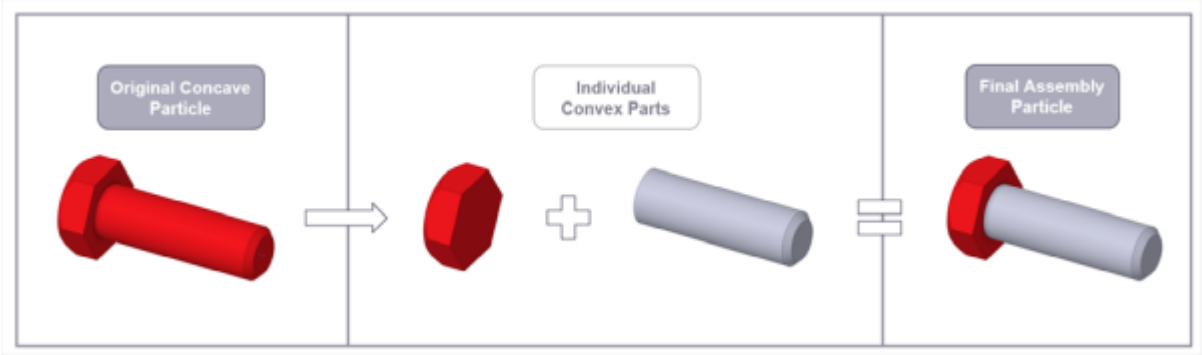
ABOUT PARTICLE ASSEMBLIES

An Assembly (p. 1389) is a type of Particle set (p. 1393) made by combining the shapes of other Particle sets to form a single, unique particle. It can be useful in cases where you are trying to replicate the glued-sphere approach common in other DEM programs, but its use also leads to faster simulations when compared to using a similar custom (imported) .stl shape. (See also About Defining and Importing Custom Particle Shapes (p. 670).)

Especially when using multiple convex shapes to represent a traditional concave shape (Figure 1), the computational savings of using an Assembly particle can be significant.

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

Figure 3.131: Traditional (imported) concave bolt shape (left) compared with the same shape defined as an Assembly particle (right)



About Assembly Parts

In Rocky, the shapes you combine together to form the Assembly shape are called parts (p. 1393). Parts are based upon Particle sets (p. 1393) that you define separately from the Assembly itself. Once included in an Assembly, the Size, Shape, Material, and Center of Mass criteria of the original Particle set making up the part are used; all other settings (Orientation, Movement, Color, etc.) are ignored by the Assembly. This means that you can define your Assembly to be heterogeneous—by specifying different materials for each part—or homogenous by specifying same material for each part.

Once the parts are defined, the whole Assembly shape will take on its own PSD (p. 1394), Orientation, Movement, color, and other settings, and will behave in the simulation just like any other individual Particle set, including being able to be used in Thermal, Coarse-Graining, and CFD Coupled simulations. (For a full list of capabilities, refer to the Rocky Particle Shapes and Capabilities (p. 41) topic.)

About Setting Custom Properties

In this version of Rocky, you are able to override the settings Rocky calculates for the Assembly's mass, area, volume, and other properties. This is usually necessary only if two or more parts in your Assembly particle overlap, and you require exact properties for your simulation to be successful.

To specify **Custom Properties** for your assembly shape, please do the following:

1. From the CAD program with which you designed your ideal (Assembly) particle shape, note the Area, Volume, Mass, Geometric Center, Center of Mass, and other properties you want Rocky to use for your Assembly particle shape.

2. In Rocky, from your Assembly particle's **Particle | Custom Properties** tab, enable the **Change Assembly Properties** checkbox, and then enter the information you want Rocky to use.

Tip:

You can use the colored dots in the **Particles Details** window to preview where your Center of Mass (blue dot) and Geometric Center (yellow dot) are located.

Assembly Mass and Geometric Centers

In this version of Rocky, the Geometric Center for an Assembly particle is represented in the Particles Details window as a yellow dot, and the Center of Mass is represented by a blue dot. The Geometric

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

Center and Center of Mass coincide when the density is uniform throughout the part. By default, Rocky determines the location for these centers based upon the following criteria:

- The shape and mass of the original particle set upon which the part is based.
- The size of the part as specified by the Assembly.

Part and Assembly Limitations

For a full list of limitations, refer to the *Assembly Particle Shape Limitations* section of the Particle and Input Limitations (p. 59) topic.

About Defining an Assembly Shape

Creating an Assembly shape involves the following steps:

1. Define the individual Particle sets upon which the parts will be based.
2. Create a new Particle set defined as an **Assembly** shape.
3. Within the new Assembly Particle set, include rows on the **Shape** sub-tab for each part that defines its location, size scale, and rotation values.

(See also Create and View a Particle Assembly (p. 398).)

What would you like to do?

- Learn more About Adding and Editing Particle Sets (p. 602)
- Create and View a Particle Assembly (p. 398)

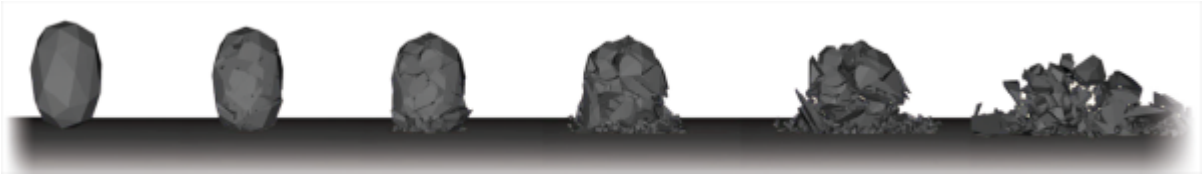
See Also:

- About Defining and Importing Custom Particle Shapes (p. 670)

ABOUT PARTICLE BREAKAGE

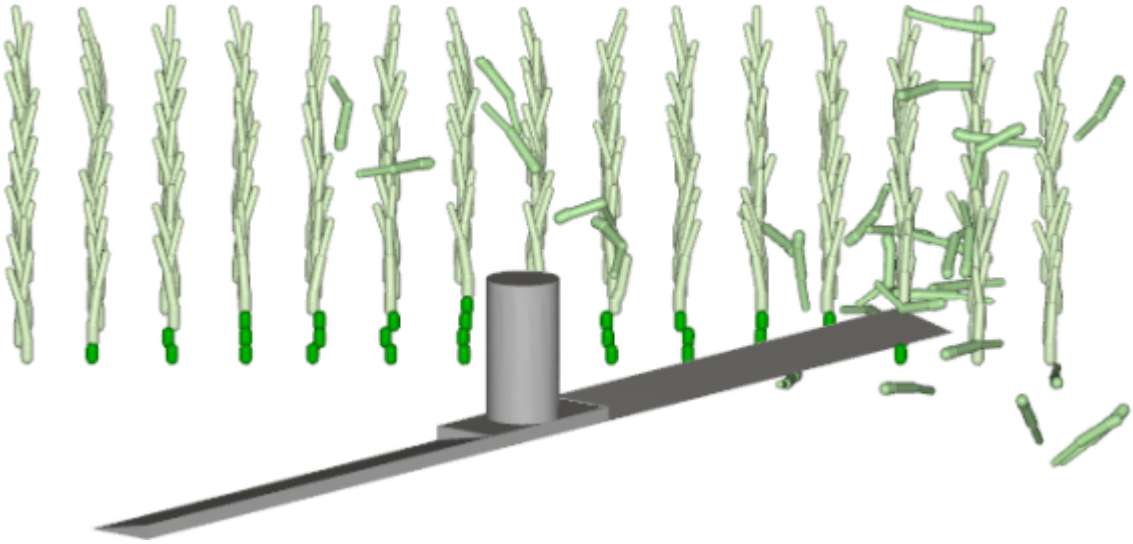
If you want your particles to break into fragments (Instantaneous Breakage (p. 1392); Figure 1) or come apart at their joints (p. 1392) (Discrete Breakage (p. 1390); Figure 2) upon contact with other particles or boundaries, you may set up your simulation to calculate particle breakage.

Figure 3.132: Example of Instantaneous Breakage in Rocky



Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

Figure 3.133: Example of Discrete Breakage in Rocky



Particle Shape, Composition, and Breakage

The Particle **Shape** type you choose and its **Composition** (see also About Adding and Editing Particle Sets (p. 602)) determines whether or not you are allowed to enable breakage calculations in Rocky, and which kinds of breakage models are available for use.

RIGID, SINGLE-ELEMENT PARTICLES AND INSTANTANEOUS BREAKAGE

For rigid particles composed of only a Single *Element* (p. 1390), only sharp-cornered (p. 1395) *Solid* (p. 1395) shapes, such as Polyhedron, Briquette, Faceted Cylinder, and non-concave Custom Polyhedron (imported) particle shapes are compatible with Instantaneous Breakage calculations. The Instantaneous Breakage models available for you to use with these kinds of particle shapes are Ab-T10 and Tavares.

All other rigid, single-element particles are incompatible with breakage models. This includes spherical (p. 1395) (Sphere) and rounded-corner (p. 1394) Solid shapes (Sphero-Cylinder, Sphero-Polygon, Sphero-Polyhedron), concave-shaped Custom Polyhedrons, and any rigid *Fiber* (p. 1391) and *Shell* (p. 1395) that are composed of only a single element.

FLEXIBLE, MULTI-ELEMENT PARTICLES AND DISCRETE BREAKAGE

In this version of Rocky, any flexible particle composed of Multiple *Element* (p. 1390) (also known as *Meshed* (p. 1392)) are compatible with Discrete Breakage calculations. For *Solid* (p. 1395) shapes, this includes multi-element Polyhedrons and both multi-element convex and concave Custom Polyhedron (imported) particle shapes. For flexible *Fiber* (p. 1391) and *Shell* (p. 1395), this includes any shape as long as they are composed of Multiple Elements.

The Discrete Breakage models available for you to use with flexible particle shapes include Griffith Energy Criterion, Shear Stress Criterion, Tensile Stress Criterion, Tensile or Shear Stress Criterion, and von Mises Stress Criterion.

Breakage Models

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

In this version of Rocky, there are Instantaneous Breakage models available for rigid particles composed of only a Single *Element* (p. 1390), and Discrete Breakage models that are specific for flexible particles composed of Multiple Elements. See below for details.

Note:

In this version of Rocky, the **TBM** breakage model is provided as an external module (p. 1391). Refer to the Install an External Module (p. 223) topic for details.

INSTANTANEOUS BREAKAGE MODELS

For single-element particles, the Instantaneous Breakage models available are the **Ab-T10** and **Tavares** models. All three models can preserve both mass and volume.

Ab-T10 Breakage Model The Ab-T10 breakage model is based upon the Voronoi fracture particle subdivision algorithm and treats every particle as a single entity that can be broken into fragments (p. 1393) instantaneously based upon the breakage force and/or energy values defined.

For more information about how the Ab-T10 breakage model works, refer to the following resources:

- The DEM Technical Manual (from the **Help** menu, point to **Manuals** and then click **DEM Technical Manual**)

Tip:

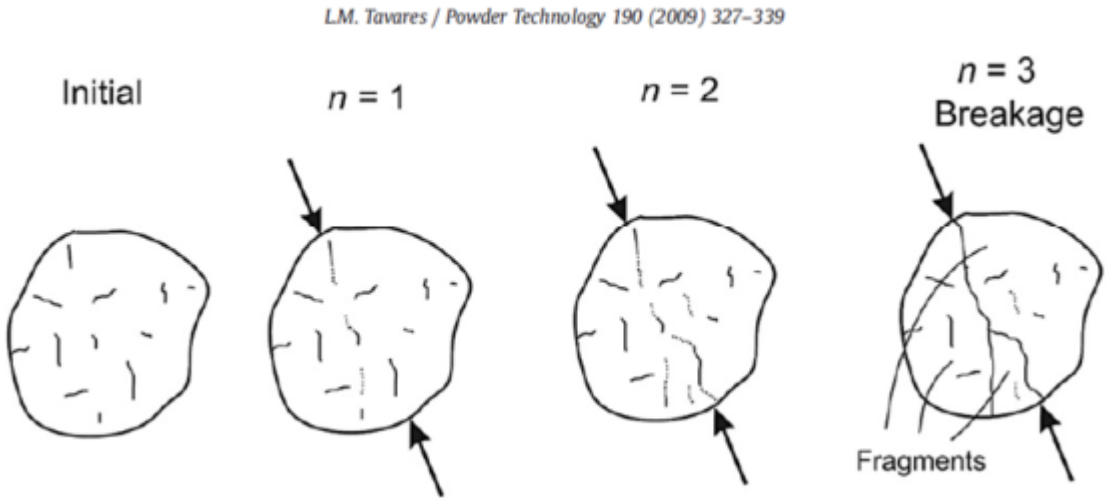
For walk-through examples of particle breakage using the Ab-T10 model, review the following resources:

- Tutorial - Drop Weight Test in the *Rocky Tutorial Guide*
- Tutorial - High Pressure Grinding Roll (HPGR) in the *Rocky Tutorial Guide*
- Tutorial - Cone Crusher in the *Rocky Tutorial Guide*

Tavares Breakage Model Like the Ab-T10 model, the Tavares breakage model is based upon the Voronoi fracture particle subdivision algorithm. In this model, however, the breakage energy probability and resulting fragment size distribution are based upon the Tavares et al. (UFRJ) approach. Modeling fracture by low-energy stressing is relevant for the simulation of a number of unit operations in particulate materials processing and handling, where particles are often subject to a complex series of loading events. Tavares' breakage model has been able to describe the progressive growth of crack-like damage (Figure 2) that ultimately leads to the fracture of a particle under stresses significantly lower than those required for breakage in a first event.

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

Figure 3.134: Illustration showing the effect of weakening due to accrual of damage in repeated loading events



For more information about how the Tavares breakage model works, refer to the following resources:

- The DEM Technical Manual (from the **Help** menu, point to **Manuals** and then click **DEM Technical Manual**)

DISCRETE BREAKAGE MODELS

Discrete breakage models can be used only for multi-element (flexible) particles. These types of particles can break only at the joints (p. 1392) that otherwise keep the Elements (p. 1390) of the particle glued together. Therefore, the fragments resulting from these breakage events will be single Elements or portions of joined Elements.

The Discrete Breakage models available in this version of Rocky are listed below:

- **Griffith Energy Criterion**: This criterion is based on an energy balance that involves the surface energy as a intrinsic property of the joints. This criterion is recommended especially for modeling breakage of flexible solid particles.
- **Shear Stress Criterion**: In this criterion, when the actual shear stress acting on a joint is larger than the specified shear stress limit, the joint will break. This is useful for modeling flexible breakage situations in which the shear or torsion loading conditions are dominant.
- **Tensile Stress Criterion**: In this criterion, when the actual tensile stress acting on a joint is larger than the specified tensile stress limit, the joint will break. This is a useful for modeling breakage of flexible particles subjected mainly to stretching or bending.
- **Tensile or Shear Stress Criterion**: In this criterion, both the actual tensile stress and the actual shear stress acting on a joint are evaluated at the same time and whichever criteria is satisfied first determines when the joint will break. This is useful for modeling breakage in cases in which both the normal and shear stresses are important.

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

- **von Mises Stress Criterion**: In this criterion, when the (combined) actual tensile stress and actual shear stress acting on a joint is larger than the von Mises stress limit, the joint will break. This is useful for modeling breakage of flexible particles under complex loading conditions.

Tip:

Refer also to the DEM Technical Manual. (From the **Help** menu, point to **Manuals** and then click **DEM Technical Manual**.)

Warnings and Limitations When Using Breakage

Due to the additional calculations involved, a simulation with breakage can be expected to take significantly longer to process. For breakage simulations with very long estimated processing times, consider using Energy Spectra instead. (See also Collecting Data on Contacts and Energy Spectra (p. 350).)

Limitations: Breakage calculations are not currently compatible with the following settings and functions:

- Coarse Grain Modeling. (See also About Physics Parameters (p. 417).)
- Any spherical or rounded-corner *Solid* (p. 1395) particle shapes, including Spheres, Sphero-Cylinder, Sphero-Polygon, and Sphero-Polyhedron particle shapes. (See also About Adding and Editing Particle Sets (p. 602).)
- Any imported Custom Polyhedron particle shapes that are concave and composed of only a single element. (See also About Defining and Importing Custom Particle Shapes (p. 670).)
- Any *Shell* (p. 1395) particle shapes composed of only a single element.
- Any *Fiber* (p. 1391) particle shapes composed of only a single element.
- For projects processed in older versions (prior to Rocky v4): - Particles Time Selection. (See also About the Particles Time Selection User Process (p. 1221).) - Particle Tagging, including Divisions Tagging. (See also About Particle Tagging (p. 1248).) - Residence Time Property. (See also About Properties (p. 933).)
- If want to save for restart purposes a partially processed simulation with breakage enabled, be aware that any fragments (p. 1393) that were in the simulation at the time you save the copy will not be present in the copied version due to incompatibilities between the features. (See also Save a Copy of a Partially Processed Simulation for Restart Purposes (p. 855).) Note: If you choose to restart processing a copy of a breakage simulation with active fragments at the time of saving, even though no fragments will be present at the initial output time, breakage should work as usual for the new Outputs that are processed thereafter.
- When analyzing the Particle Properties of your completed breakage simulation (see also About Properties (p. 933)), be aware that the Impact Specific Energy values you display for particles that are capable of breaking may be inaccurate. This is because Rocky treats differently the storage of energy values for these types of particles only. Note also that this exception does not include broken fragments (p. 1393) that are close to the minimum size, as these are considered non-breakable and, therefore, will display the correct Impact Specific Energy values.

What would you like to do?

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

- Enable and View Particle Breakage (p. 397)
- Learn more About Adding and Editing Particle Sets (p. 602)

ABOUT DEFINING AND IMPORTING CUSTOM PARTICLE SHAPES

If the existing particle shapes that come with Rocky do not meet your needs, you can define and import custom particle shapes to use within your simulations. For example, you can achieve a model of a real-world object by using a 3D scanner, or you can design one using a CAD tool like Ansys SpaceClaim. Nearly any kind of shape can be supported in Rocky: convex, concave, perforated, closed, open, and more.

About Custom Particle Shape Types

There are three categories of custom particle shapes that can be imported into Rocky:

- Custom *Fiber* (p. 1391) particles, which are defined in a text or spreadsheet file.
- Custom *Shell* (p. 1395) particles, which are defined in an .stl file.
- Custom Polyhedron (*Solid* (p. 1395)) particles, which are defined in an .stl file, and can be categorized as either *Convex* (p. 1390) or *Concave* (p. 1390). It is these Solid particles that are the best candidates for scanning using a 3D scanner.

Note:

Different shapes types have different capabilities in Rocky. Refer to Rocky Particle Shapes and Capabilities (p. 41) for specifics.

See also About Adding and Editing Particle Sets (p. 602).

About Designing Custom Shell and Polyhedron Shapes using a CAD Program

To import Custom *Shell* (p. 1395) and Custom Polyhedron (*Solid* (p. 1395)) particles, Rocky needs an .stl file. One way to get that .stl file is to use a CAD program, such as Ansys SpaceClaim, to define your shape and then save the geometry as an .stl file. (See also Appendix F: Save Geometries as XGL, STL, or DXF Files (p. 1307).)

Tip:

It is recommended that especially for Shell particle shapes, you do not re-import into Rocky rendered shapes (p. 1394) that have been previously exported out of Rocky. (See also Export a Particle Shape to an STL File (p. 698).) This is because after you import your original Shell shape, Rocky turns that 2D shape into a 3D shape in order to calculate the thickness value, and this 3D aspect is retained when you export the rendered shape out of Rocky. Because Rocky will except only an empty 2D shape when importing Shells, it will not recognize the (now 3D) exported shape as such and will instead want to import it as a solid 3D shape. To avoid this issue, export the **Imported Shape** instead.

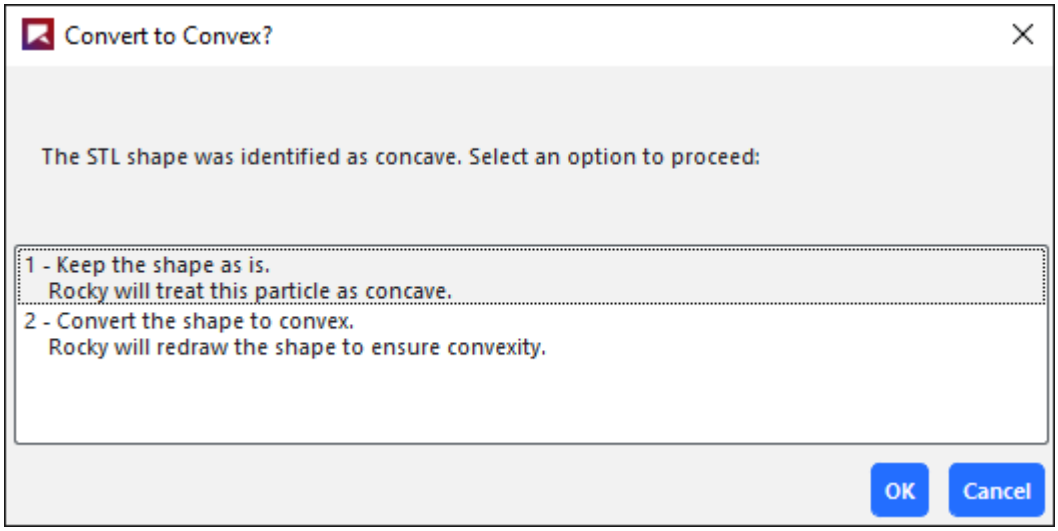
About Importing Convex and Concave Polyhedron Shapes

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

How you design your Custom Polyhedron shape determines whether Rocky will categorize it as a *Convex* (p. 1390) or *Concave* (p. 1390) shape after import. Convex shapes will have no dents or hollows formed by its triangle faces. If Rocky detects even the smallest dent or hollow in the imported .stl, it will categorize that shape as concave-whether that was your intention or not.

Rocky will notify you of this detection via a **Convert to Convex?** message (Figure 1) that will display after import.

Figure 3.135: Convert to Convex message you might see after importing a Custom Polyhedron shape



If you are satisfied with the shape being treated as concave, then you can choose the first option on the **Convert to Convex?** message.

If you still want your shape to be treated as convex, however, you will either need to have Rocky re-draw the shape so that it meets the convexity requirements, or you will need to re-draw and re-import the shape yourself. To do the former, you simply need to select the second option on the **Convert to Convex?** message. Rocky will use a convex hull algorithm to convert the shape for you.

Tips:

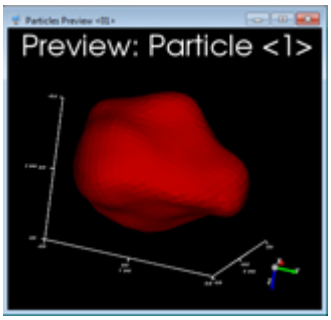
- To see a walk-through example of importing a Custom Polyhedron particle shape, review Workshop 09 - Tablet Coater.
- To learn more about how Rocky categorizes and calculates convex and concave shapes, refer to the DEM Technical Manual. (From the Rocky **Help** menu, point to **Manuals**, and then click **DEM Technical Manual**.)

About Scanning 3D Solid Particle Shapes

Besides designing your Custom Polyhedron particle shape in a CAD program, you can also use a 3D scanner to convert your real-life object into an .stl file that you can then import into Rocky, as shown in Table 1.

Table 1: Typical scenario for scanning a real-world particle shape and importing it into Rocky

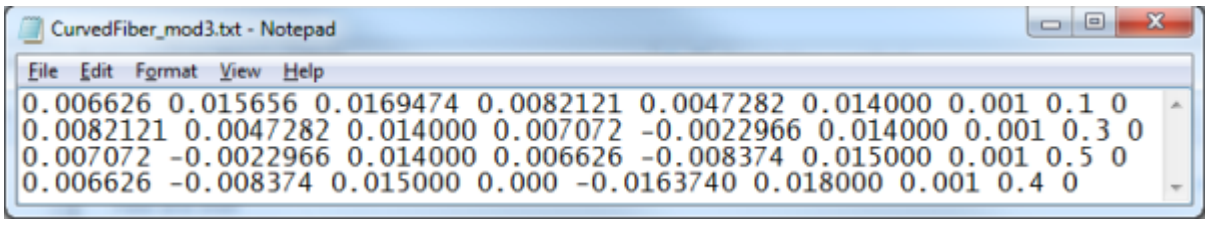
Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

			
1. A real-world particle-such as this rock-is scanned on a 3D scanner	2. The scanned particle is saved as a 3D model (.stl file)	3. The .stl file is imported into Rocky as a **Custom Polyhedron** particle shape	4. A Rocky simulation is conducted using the new custom particle shape

About Creating a Custom Fiber Definition File

Unlike Custom *Shell* (p. 1395) and Custom Polyhedron (*Solid* (p. 1395)) shapes, Custom *Fiber* (p. 1391) shapes are defined in a text file (Figure 2) or spreadsheet file (Figure 3) (see also File Types and Folders in Rocky (p. 129)).

Figure 3.136: Example Custom Fiber defined in a text file



Fiber shapes are made up of separate but connected *Segment* (p. 1394). The location and characteristics of each Segment are defined in the Custom Fiber's definition file, which contains:

- the coordinate location of the Segment's starting and ending nodes
- the Segment's diameter (in meters)
- an (optional) multiplier of the Young's modulus for the Segment, which is used to define a custom fiber with nonuniform flexibility
- an (optional) parameter for turning the Fiber into a frozen Fiber (p. 1391). This works by defining one or more "frozen" Segments, which when used for multi-element (flexible and/or discrete-breakable) particles (see also About Adding and Editing Particle Sets (p. 602)) combined with Custom Inputs (see also About Adding and Editing Particle Inputs (p. 708)), allows the frozen portion of the fiber to "stick" to a certain location and have only the unfrozen portions of the fiber respond to interactions from other objects in the simulation. This can be useful for simulating a field of individual grass blades, for example. Important: This feature works only when the Custom Fiber is composed of multiple elements. (The Particle set's (p. 1393) **Composition** parameter must be set to **Multiple Elements**.)

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

Each individual Segment making up the Fiber shape is defined in a separate row according to the following format:

`x1,y1,z1,x2,y2,z2,diameter,k_multiplier,frozen,tag`

Where:

- `(x1,y1,z1)` represent the Segment's starting nodal coordinate location (in meters)
- `(x2,y2,z2)` represent the Segment's ending nodal coordinate location (in meters)
- `diameter` is the diameter (in meters) of the indicated Segment
- (optional) `k_multiplier` is the Young's modulus multiplier for the indicated Segment Note: This Young's modulus multiplier is referred to as **M** in the DEM Technical Manual. (From the **Help** menu, point to **Manuals**, and then click **DEM Technical Manual**.)
- (optional) `frozen` indicates whether Segment will be treated as frozen (1) or unfrozen (0)
- (Modules Only) tag: this column will only be used with modules that models the custom joint behavior.

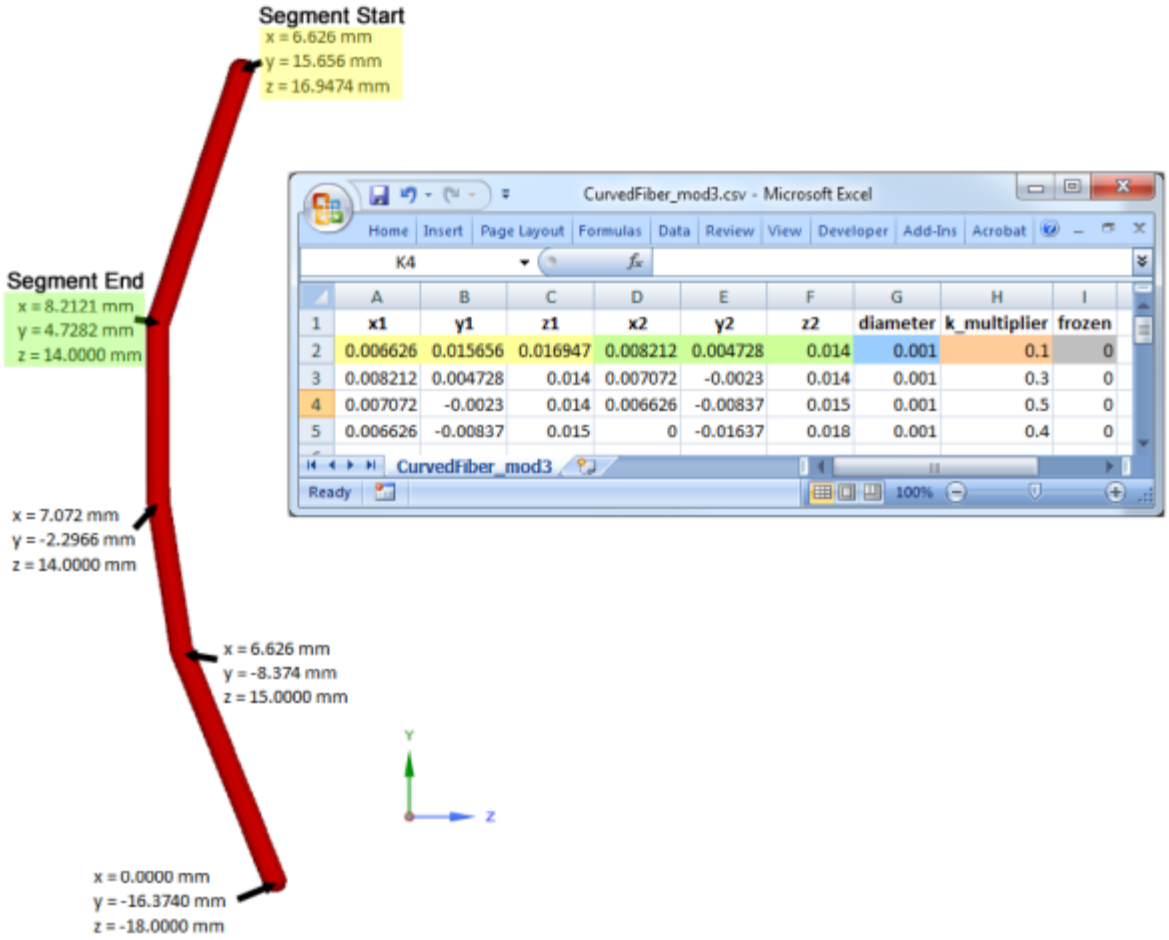
Notes:

- Values can be separated by spaces (text files only; Figure 2) or commas (spreadsheet files only; Figure 3).
- For spreadsheet files only, the first row can be a header row, but this is not required (Figure 3).
- Any lines that begin with a number sign (#) (also known as a pound sign or hashtag) will be ignored.
- If left undefined, Rocky will treat `k_multiplier` as 1 and `frozen` as 0.

For example, to create the four-segmented fiber in Figure 3, the coordinate location for the node that starts the first Segment is entered on the row beneath the header (yellow), followed immediately by the coordinate location of the node that ends the first Segment (green), followed by its equivalent diameter (blue)-all in meters-followed by the Young's modulus multiplier (orange), and frozen indicator (gray).

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

Figure 3.137: Example four-Segmented fiber and the .csv file used to define it



	A	B	C	D	E	F	G	H	I
1	x1	y1	z1	x2	y2	z2	diameter	k_multiplier	frozen
2	0.006626	0.015656	0.016947	0.008212	0.004728	0.014	0.001	0.1	0
3	0.008212	0.004728	0.014	0.007072	-0.0023	0.014	0.001	0.3	0
4	0.007072	-0.0023	0.014	0.006626	-0.00837	0.015	0.001	0.5	0
5	0.006626	-0.00837	0.015	0	-0.01637	0.018	0.001	0.4	0

Figure 3.138: Example .csv file used with modules that models the custom joint behavior

x1	y1	z1	x2	y2	z2	diameter	k_multiplier	frozen	tag
0	0	-1	0	0	0	0.2	1	0	2
0	0	0	0	0	1	0.2	1	0	2
0	0	1	0	0	2	0.2	1	1	3
0	0	2	0	0	3	0.2	1	0	1
0	0	3	0	0	4	0.2	1	0	1

The third row defines the starting and ending node locations for the second Segment in the fiber, and so on. In this way, you can create a wide variety of different segmented shapes.

Important:

To ensure predictable results when simulating a custom fiber with Segments that cross, design the fiber so that it includes a node at the desired crossing point. For example, when designing a + (plus) shaped particle whose two Segments cross at their center points, ensure that you include a node in the center of the shape by creating four separate entries in your definition file rather than two.

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

See also the *Flexible custom fibers* section in the DEM Technical Manual. (From the **Help** menu, point to **Manuals** and then click **DEM Technical Manual**.)

Tips and Limitations for Designing and Importing Custom Particle Shapes

---

Important:

The complexity of the shape you import has an impact on the overall simulation time so it is important that you aim for as few triangles (or, in the case of Custom Fibers, nodes) in your design as possible that still results in a uniform mesh with good shape representation (see Figure 4). The finer the mesh of your shape, the longer it will take for your simulation to complete.

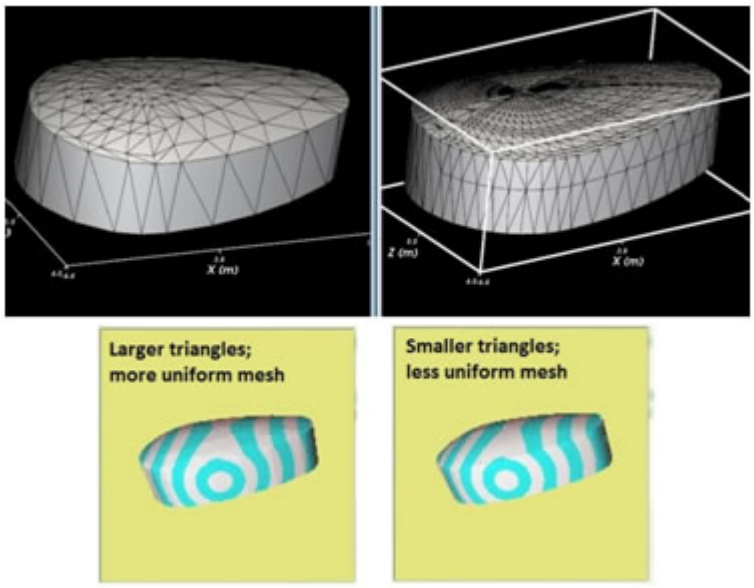


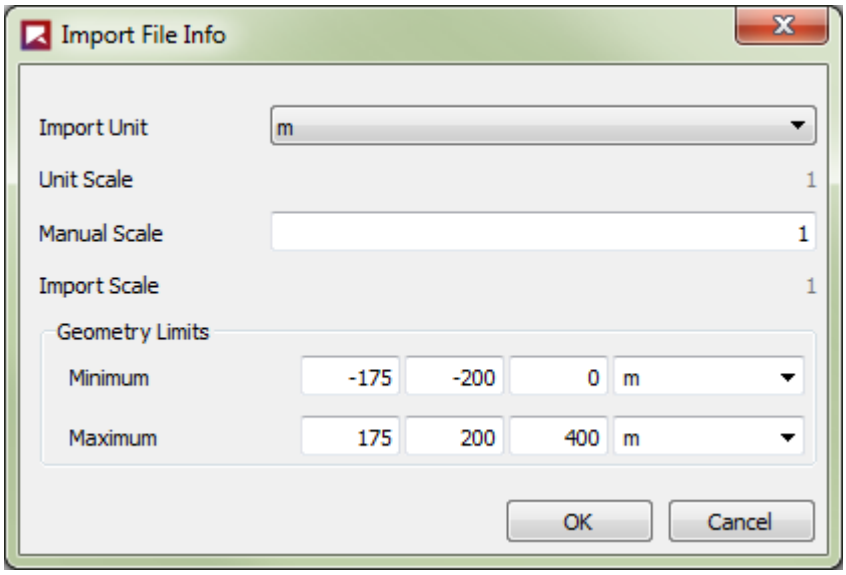
Figure 4 : Shape mesh comparison: The shape on the left has good meshing and will process faster.

About Importing a Custom Particle Shape

When you choose to import a custom particle shape, you will first be asked to select the shape file you want to use, and will then be asked to verify the import units and scaling criteria (Figure 4).

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

Figure 3.139: Import File Info dialog



By default, Rocky will import the file as the original size specified in the .stl or .txt file but you can scale the particle down or up as you wish. Note: How you choose to size and scale your particle shape upon import affects how the particle size will be displayed in the **Particles Details** window only when **Original Size Scale** is selected on the **Size** tab for **Size Type**.

Use the table below to understand the various import parameters you can set for your custom particle shapes.

Table 1: Options on the Import File Info dialog

Setting	Description	Range
Import Unit	Enables you to change the units of the imported shape.	Various units of length
Unit Scale	Displays the unit scale based upon the **Import Unit** set. For example, if **Import Unit** is left as the default value, the Unit Scale will be 1.	Automatically determined
Manual Scale	Enables you to manually adjust the scale by any factor you want. Leaving the value at 1 will have no additional affect upon the scale.	All values
Import Scale	Displays the final import scale based upon the **Manual Scale** and **Import Unit** values set. For example, if both those options are left as the default values, the Import Scale will be 1.	Automatically determined
Geometry Limits		
Minimum	The minimum coordinates (in X Y Z format) of the particle's geometry.	No limit
Maximum	The maximum coordinates (in X Y Z format) of the particle's geometry.	No limit

What would you like to do?

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

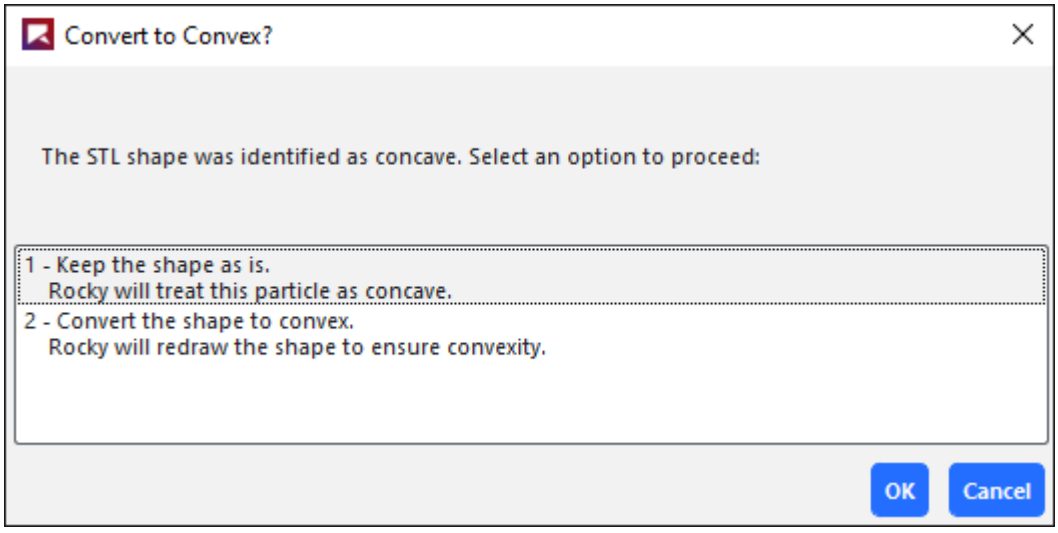
- Import a Custom Particle Shape (p. 677)
- Learn more About Adding and Editing Particle Sets (p. 602)
- Learn more about Rocky Particle Shapes and Capabilities (p. 41)

See Also:

- A fiber or other particle shape disappeared from my simulation during processing (p. 1357)
- I get an "Unsupported Particle" error when I open an older project (p. 1329)

IMPORT A CUSTOM PARTICLE SHAPE

1. From the **Data** panel, right-click **Particles**, and then click **Create Particle**. A new Particle set is listed under **Particles** in alphabetical (case-sensitive) order.
2. Select the Particle set you just added, and then from the **Data Editors** panel, find the **Shape** list and do one of the following:
   - To import a custom *Fiber* (p. 1391) shape from a text or spreadsheet file, click **Custom Fiber**.
   - To import a custom *Shell* (p. 1395) shape from an .stl file, click **Custom Shell**.
   - To import a custom *Solid* (p. 1395) shape from an .stl file, click **Custom Polyhedron**.
3. From the **Select file to import** dialog, locate and select the file you want to use, and then click **Open**.
4. From the **Import File Info** dialog, choose the unit and scaling options you want, verify the resulting **Geometry Limits**, and then click **OK**.
5. If you are importing a Custom Polyhedron shape and happen to see the following **Convert to Convex?** message, select one of the following options, and then click **OK**:



   - If you want Rocky to calculate the shape as a concave shape, select the **1 - Keep the shape as is** option.

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

- If you want Rocky to re-draw your shape so that it qualifies as convex and can be calculated as such, select the **2 - Convert the shape to convex** option.

6. Click the **View** button to see a 3D view of the largest particle in the *Particle set* (p. 1393) in a Particle Details window. Tip:Use the mouse to rotate, pan, and zoom the image, just like you would in a 3D View window. (See also Use the Mouse, Keyboard, or Toolbar to Change a 3D View (p. 1047).) Also, use the options on the Coloring tab to change the options for Edges and/or Faces (see also About Using the Coloring Tab to Change a 3D View (p. 1074).)

7. Continue to set up your Particle sets as you normally would. (See also Add and Edit Particle Sets (p. 602).)

Note:

To choose a different file to use for your Particle set, from the **Shape** sub-tab click **Load File**, and then follow steps 3-7 above.

Tip:

It is recommended that especially for Shell particle shapes, you do not re-import into Rocky rendered shapes (p. 1394) that have been previously exported out of Rocky. (See also Export a Particle Shape to an STL File (p. 698).) This is because after you import your original Shell shape, Rocky turns that 2D shape into a 3D shape in order to calculate the thickness value, and this 3D aspect is retained when you export the rendered shape out of Rocky. Because Rocky will except only an empty 2D shape when importing Shells, it will not recognize the (now 3D) exported shape as such and will instead want to import it as a solid 3D shape. To resolve this issue, export the **Imported Shape** instead.

See Also:

- About Defining and Importing Custom Particle Shapes (p. 670)
- About Adding and Editing Particle Sets (p. 602)
- A fiber or other particle shape disappeared from my simulation during processing (p. 1357)
- I get an "Unsupported Particle" error when I open an older project (p. 1329)

ADD A NEW PARTICLE SET

- From the **Data** panel, right-click **Particles**, and then click **Create Particle**. A new Particle set is listed under **Particles** in alphabetical (case-sensitive) order.

See Also:

- About Adding and Editing Particle Sets (p. 602)
- Modify the Parameters of a Particle Set (p. 678)
- Remove an Existing Particle Set (p. 698)

MODIFY THE PARAMETERS OF A PARTICLE SET

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

1. From the **Data** panel, under **Particles**, click the name of the Particle set you want to modify. The parameters for the Particle set become active in the **Data Editors** panel.
2. From the **Data Editors** panel, modify the parameters as you want.

Tip:

To see a detailed 3D view of your particle shape, click the **View** button.

See Also:

- About Adding and Editing Particle Sets (p. 602)
- Add a New Particle Set (p. 678)
- Remove an Existing Particle Set (p. 698)
- Export a Particle Shape to an STL File (p. 698)

CREATE AND MODIFY A PARTICLES DETAILS WINDOW

A Particles Details window is where you preview the shape you defined in your Particle set (p. 1393). If you have set collision statistics to be collected during processing, it is also where you will view the details of this data on a representative particle.

Like other types of Rocky windows, you are able to define how the window and the items within it appear and function on screen. After you set up your Particles Details windows, you can choose to reuse the zoom, rotation, and pan settings in another window by saving and applying a Custom Camera Preset. You can also use the Particles Details window as a basis for creating an animation of the Intra-particle Collision Statistics that were collected during the simulation.

What do you want to do?

- Learn more About Particles Details Windows (p. 679)
- Preview a Particle Shape in 3D (p. 681)
- Change a Particles Details Window (p. 682)
- Save a View with Custom Cameras (p. 1093)
- Learn more About Adding and Editing Particle Sets (p. 602)
- Create and Save an Animation (p. 1166)

See Also:

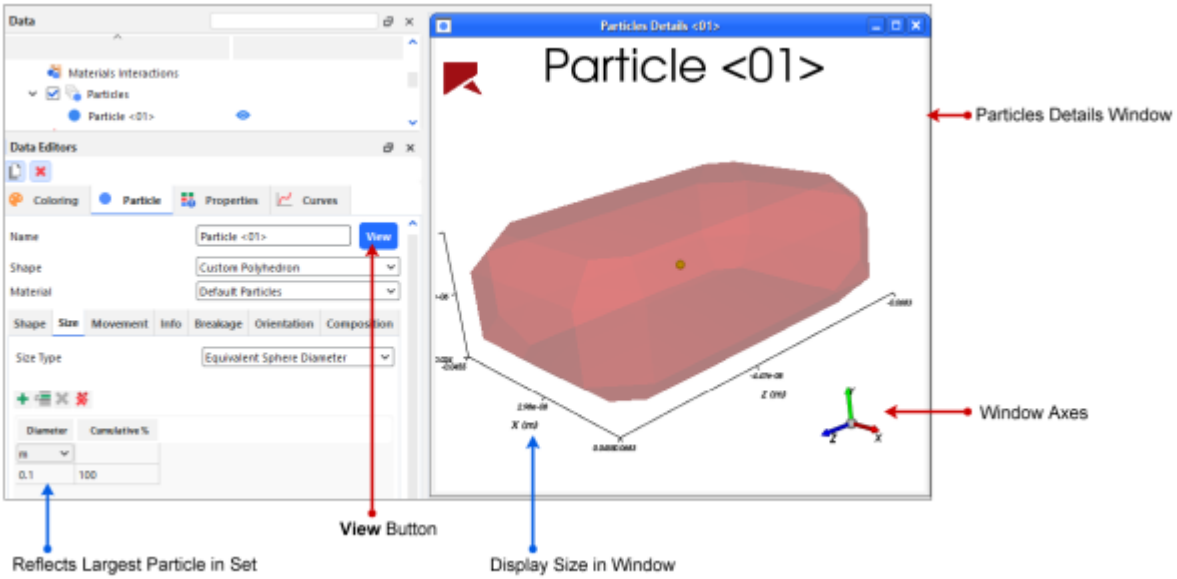
- Add and Edit Particle Sets (p. 602)
- About Intra-Particle Collision Statistics (p. 324)
- Enable and View Collision Statistics for Particle Surfaces (Intra) (p. 346)

About Particles Details Windows

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

A Particles Details window is where you preview the shape you defined in your Particle set (p. 1393) (Figure 1).

Figure 3.140: Particles Details window overview when used to preview a particle shape

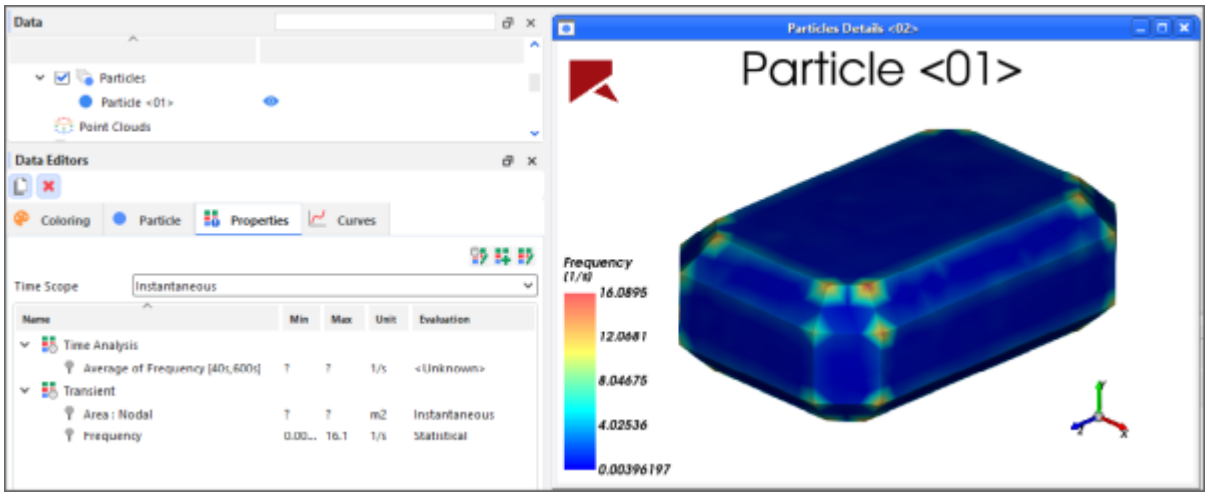


Note:

The Geometric Center for a particle is represented in the Particles Details window as a yellow dot.

In addition, if you have set collision statistics to be collected during processing (see also About Collision Statistics for Particles (p. 324)), it is also where you will view the details of this data on a representative particle (Figure 2).

Figure 3.141: Particles Details window overview when used to view collision statistics after processing



Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

As with 3D View windows, you can use the data shown in the Particles Details window as a basis for creating an animation of the collisions results.

In both cases, you can open this window from the **View** button on the **Particles** tab of the selected Particle set (Figure 1).

In this version of Rocky, the preview reflects the size of the largest particle in the selected Particle set. (See also the *Size Tab* section of the About Adding and Editing Particle Sets (p. 602).)

Notes:

- In order to see the collision statistics, please remove the transparency of the particle.
- When displaying particles at a very small scale, there can be display issues related to rendering and lighting. (See also Parts of my particle disappear or appear cut off (p. 1324).)
- As with other windows in Rocky, there are various ways you can change what appears in a Particles Details window, including fonts, overlays, background colors, and grid lines.

What do you want to do?

- Preview a Particle Shape in 3D (p. 681)
- Change a Particles Details Window (p. 682)
- Learn more About Adding and Editing Particle Sets (p. 602)
- Learn more About Intra-Particle Collision Statistics (p. 324)
- Create and Save an Animation (p. 1166)

Preview a Particle Shape in 3D

1. From the **Data** panel, under **Particles**, click the name of the Particle set you want to preview. The parameters for the Particle set become active in the **Data Editors** panel.
2. From the **Data Editors** panel, on the **Particle** tab, click the **View** button. A new **Particles Details** window appears in the Workspace showing the selected particle shape in 3D.

Tips:

- To show the exact size of an imported shape within the **Particle Details** window, use the **Original Scale Size** for **Size Type** and set the biggest size to 100%.
- You can change the look of the Particles Details window in ways similar to a 3D Window. (See also Change a Particles Details Window (p. 682).)

Note: The window will automatically update when the selected Particle Set's parameters change.

See Also:

- About Particles Details Windows (p. 679)
- About Adding and Editing Particle Sets (p. 602)

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

Change a Particles Details Window

There are several ways you can change what you see in the Particles Details window once you have created it, as explained below.

- Change the level of zoom or rotation of the Particle by using your mouse, keyboard, or the camera options on the visualization toolbar.
- Change the display of the Particle through the Coloring tab on the Data Editors panel.
- Change the look of the window display through the Window Editors panel, including changing the default text overlay or adding a logo or image to the display.
- When showing a Property after particle collisions have been collected, you can change the display of the colored legend by modifying the Color Scales for the Property you are displaying.
- Change the display of the window axes through the top-most "root" Particles Details category on the Windows Editors panel.

What would you like to do?

- Learn more about Using the Mouse, Keyboard, or Toolbar to Change a the View (p. 1043)
- Learn more about Using the Coloring Tab to Change a Particles Details Window (p. 694)
- Learn more about Using the Window Editors Panel to Change a Particles Details Window (p. 682)
- Learn more About Color Scales (p. 1086)
- Learn more About Using the Window Editors Panel to Change the Window Axes Displays (p. 1071)

See Also:

- Learn more About Particles Details Windows (p. 679)

USING THE WINDOW EDITORS PANEL TO CHANGE A PARTICLES DETAILS WINDOW

From the Window Editors panel, you can modify just the selected Particles Details window the change the background color, font color/sizes, text and image overlays, and synchronize the time steps displayed when two or more view windows are open. You may also adjust the axes display settings for all the windows in your project.

What would you like to do?

- Learn more About Using the Window Editors Panel to Change the Selected Particles Details Window (p. 683)
- Change Window Display and Data Update Options for the Selected Particles Details Window (p. 694)
- Add or Edit an Image Overlay for a Particles Details Window (p. 692)
- Add or Edit a Text Overlay for a Particles Details Window (p. 692)
- Learn more About Using the Window Editors Panel to Change the Window Axes Displays (p. 1071)

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

- Change the Axes Display for all Rocky Windows (p. 1074)

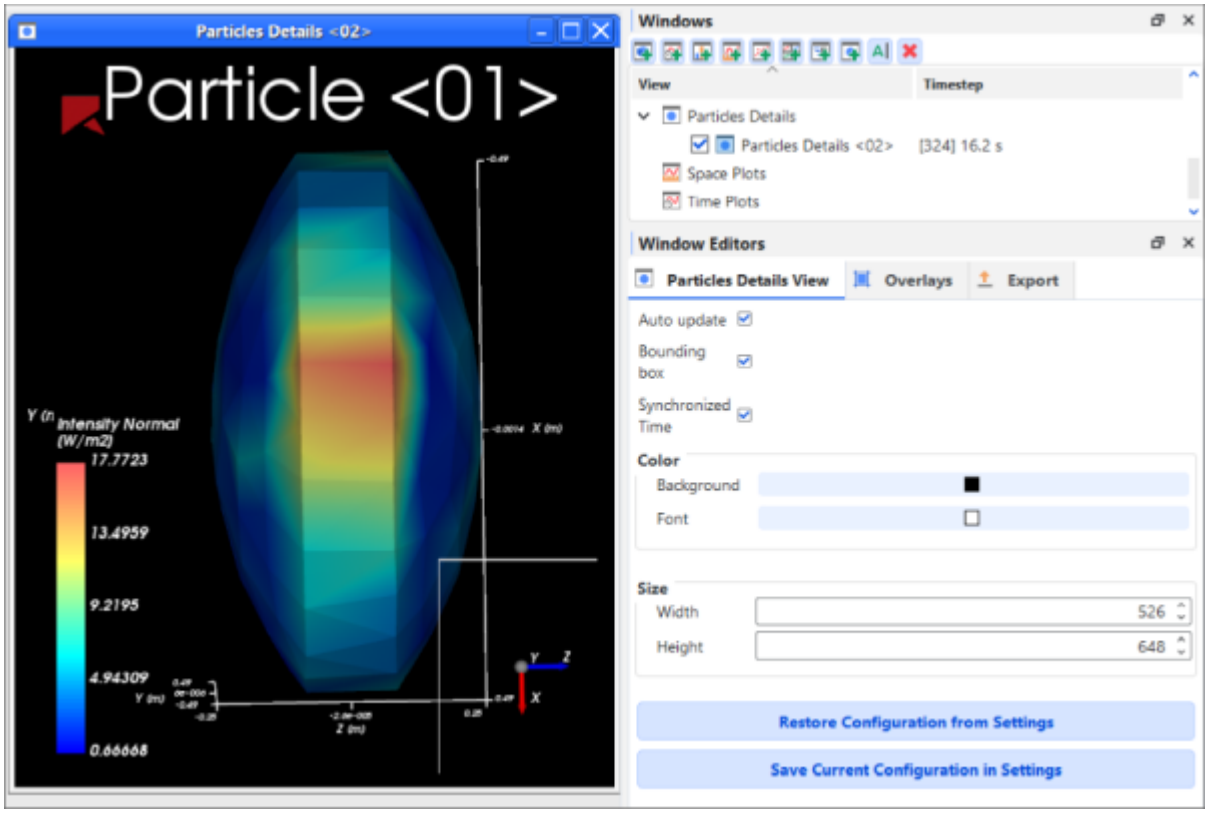
See Also:

- About Particles Details Windows (p. 679)

ABOUT USING THE WINDOW EDITORS PANEL TO CHANGE THE SELECTED PARTICLES DETAILS WINDOW

When you use the Window Editors panel to change the selected Particles Details window, you have the ability to change how items in the window appear on the screen. These items include background color, font color/sizes, and text and/or image overlays (Figure 1). You can also change how multiple windows of the same type respond to changes in time steps.

Figure 3.142: Particles Details window and Window Editors panel settings



Tips:

- You can also use Rocky Preferences to change the background color and fonts for all the Particles Details windows in your project, not just the selected one. (See the "Preferences for Windows Properties" section in the About Setting Global Preferences (p. 227) topic.)
- You can also change the window axes display for all windows in your project, including Particles Details windows. (See also About Using the Window Editors Panel to Change the Window Axes Displays (p. 1071).)

In this section, we will cover only the items on the **Particles Details View** tab and the **Overlays** tab. Note: The **Export** tab will be covered in the About Exporting Data and Images Out of Rocky (p. 1155) topic.

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

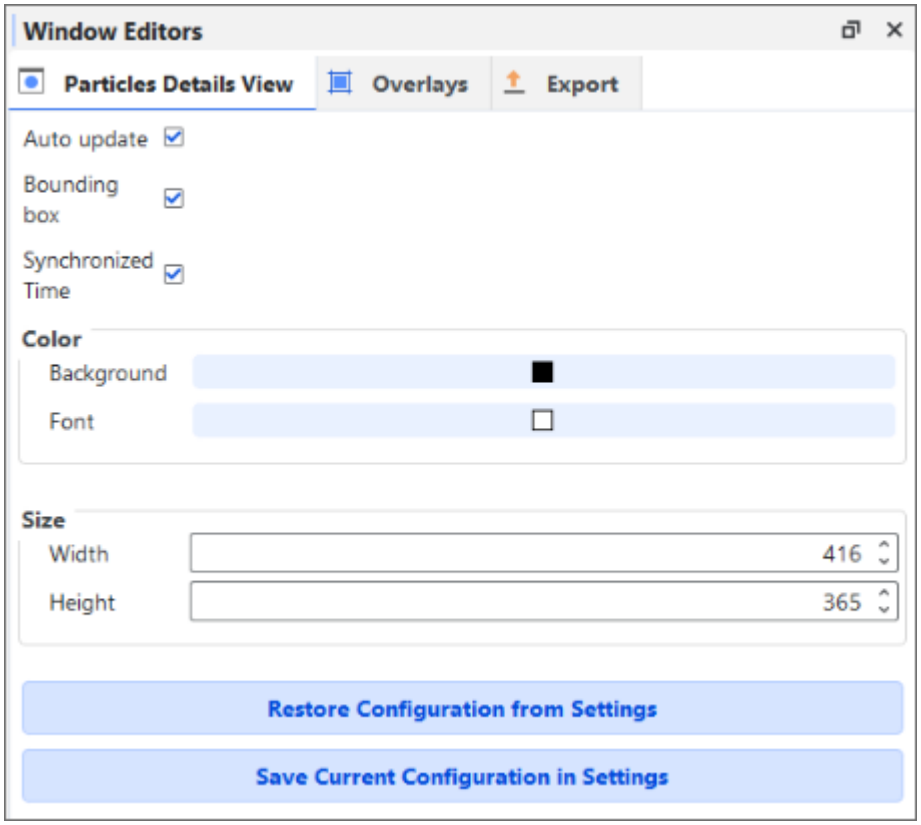
You use the **Particles Details View** tab on the **Window Editors** panel to change how the window appears and how the data displayed within it is updated. These options include changing display items like the background color; labels, lines, and axes colors; color scale label sizes; and window height and width. In addition, you can choose to show or hide the **Bounding box**, which displays measurements illustrating the simulation boundaries; use **Auto update** to enable or prevent the view from being updated when data affecting it is changed; or choose to have multiple views updated with the same time step by using **Synchronized Time**.

You use the **Overlays** tab on the **Window Editors** panel to change the default text or add image overlays. By default, the text overlay includes the name of the Particle set (p. 1393) that you are previewing in the window, but can be changed or removed. Text overlays to include your own custom text or can include Keywords that will show project data, and Input/Output variables, directly on your window view. Image overlays can include company or association logos, photos of real-world equipment, or any applicable image that compliments your simulation. Adding these kinds of overlays are especially useful for generating videos and images from your simulation.

PARTICLES DETAILS VIEW AND OVERLAYS TAB SETTING DEFINITIONS

See the images and table below to understand more about how you can use the Window Editors panel to change the selected Particles Details window.

Figure 3.143: Particles Details View tab options on the Window Editors panel

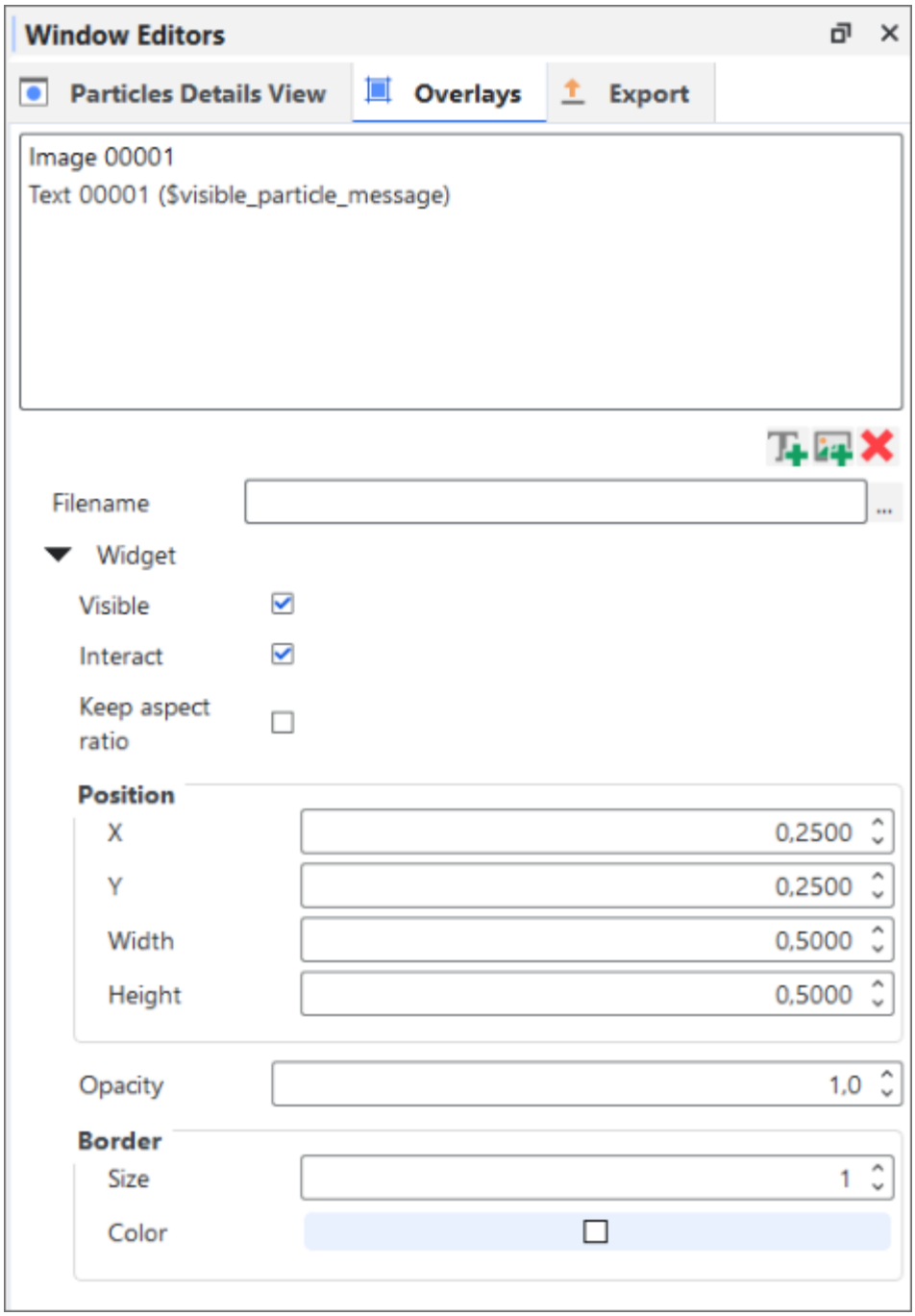


Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

Figure 3.144: Initial Overlays tab options on the Window Editors panel showing default Particle Set text

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

Figure 3.145: Image overlay options on the Window Editors panel



Particles Details View tab options on the Window Editors panel

Setting	Description	Range
Auto update	When enabled, any change made to the display settings of the selected Particles Details window or to the particle shape displayed, will be updated in the Particles Preview window automatically. Because automatically updating the window with each separate change can be computationally intensive, you can clear this option to	Turns on or off

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

Setting	Description	Range
	have all calculated items and more computationally intensive display options remain unchanged in the window until you enable **Auto update** again. **Tip:** You will know when the calculations in a Particles Details window are not being updated when you see a thick red border around the window.	
Bounding box	When enabled, displays measurements illustrating the simulation boundaries (p. 1389) in the selected Particles Details window.	Turns on or off
Synchronized Time	When enabled and the window (or any other window with this checkbox enabled) is selected, the details shown in this (and any other Synchronized Time) window will be updated when the current time step is changed on the Time toolbar. (See also About the Time Toolbar (p. 1279)).When cleared and the window is selected, only this window will be updated when the current time step is changed on the Time toolbar. **Tip:** To keep the time step synchronized between multiple **Particles Details** or other 3D windows, ensure each window has this option enabled.	Turns on or off
Color Background	Enables you to change the color that appears behind the particle shape in the Particles Details window.	Options limited by the choices in the **Select Color** dialog
Color Font	Enables you to change the color of the labels, borders, and axes lines displayed in the Particles Details window.	Options limited by the choices in the **Select Color** dialog
Size Width	Enables you to numerically change how wide the selected window is. Useful in cases where you need exact dimensions for image consistency.	Whole positive values between 123 and 5000.
Size Height	Enables you to numerically change how tall the window is. Useful in cases where you need exact dimensions for image consistency.	Whole positive values between 0 (zero) and 5000.

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

Setting	Description	Range
Restore Configuration from Settings	Clicking this button replaces the values set on the tab with the ones that have been saved to the (internal) Rocky Settings folder. (See also Rocky File Types and Folders (p. 129)).	(Button selection)
Save Current Configuration in Settings	Clicking this button overwrites the values that have been saved to the (internal) Rocky Settings folder with the ones currently set on the tab. (See also Rocky File Types and Folders (p. 129)).	(Button selection)

Overlays tab options on the Window Editors panel for Text and Image Overlays

Setting	Description	Range
Text Overlays Only		
Input	This box displays the text, phrase, or keyword that you want overlaid onto the selected window. It can be static text that you type in, can be dynamic text by using the **Keyword for** list, or can be a combination of both.	No limit
Bold	Enables the selected text overlay to appear bold in the window.	Turns on or off
Italic	Enables the selected text overlay to appear italic in the window.	Turns on or off
Scale Mode	Determines how the text in the selected overlay scales in relation to its **Font Size**, the sizing box that surrounds it, and size of the window itself. Specifically: • **Font Size** uses the value in the **Font Size** box to set the size of the text in relation to the computer screen. With this option, the text (and its sizing box) stays the same size no matter the height and width of the window that contains it. • **Fit** adjusts the size of the text based upon the sizing box that surrounds it. With this option, the text grows or shrinks in scale with the height and width of its sizing box. • **Scaled Font Size** uses the value in the **Font Size** box to set the size of the text in relation to the height and width of the window. With this option, the text grows or shrinks in scale with the size of the window.	Font Size; Fit; Scaled Font Size
Font Size	When **Font Size** or **Scaled Font Size** are chosen for **Scale Mode**, this sets the size of the text in the selected overlay.	Whole positive values

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

Setting	Description	Range
Justification	Aligns the selected text horizontally in the overlay box. **Tip:** This feature is really only useful when you several lines of text. Use hard returns (press the Enter key) to force text to the next line.	Left Center Right
Color	Enables you to indicate from what location the font color overlay text will be taken. Specifically: • **From window** uses the **Color Font** settings in the **(Window Type)** tab. (See Table 1 above.) • **Custom** enables you to choose a unique color separate from the other text used in the window.	From window Custom
Keyword for	Enables you to enter Rocky-specific placeholder values for data and settings that pertain to your simulation. As your simulation processes or as you change your time step, and assuming you have selected **Auto update** on the **(Window Type)** tab (see Table 1 above), these values will be updated automatically in the text overlay. In addition to the items explained below, if you have defined any Input or Output Variables (see also About Defining and Using Input Variables (p. 243) and About Defining Output Variables (p. 250)), these will be listed here as well. See below for more information on keywords. • **Simulation time** lists the current time step information presented in the same format as provided on the Time toolbar. This includes the number of *output files* (p. 1393) saved, the current output time value, and the time step units. For example, **(5) 0.25 s** represents the point in the simulation where 5 output files were saved after 0.25 seconds. (See also About the Time Toolbar (p. 1279)). • **Simulation time index** lists the number of time step output files that were saved at the current time step selected. • **Simulation time value** lists the particular moment of Simulation time at the current time step selected. For example, **0.25 s**. • **Study Customer Name** lists what you have entered in the **Customer Name** box on the **Study** portion of the **Data** panel. (See also Study Parameters (p. 415)).	Simulation time; Simulation time index; Simulation time value; Study Customer Name; Study Description; Study N. Particles; Study Name; Visible Particle Message

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

Setting	Description	Range
	• **Study Description** lists what you have entered in the **Description** box on the **Study** portion of the **Data** panel. (See also Study Parameters (p. 415)). • **Study N. Particles** lists the total number of whole particles that are located within the simulation boundaries at the current time step. • **Study Name** lists what you have entered in the **Study Name** box on the **Study** portion of the **Data** panel. (See also Study Parameters (p. 415)). • **Visible Particle Message** lists the name of the *Particle set* (p. 1393) currently being shown in the window. **Tip:** If you find that it takes too long to update the text displayed when changing time steps or other data, consider clearing the **Auto update** checkbox on the **(Window Type)** tab until your changes are complete. (See Table 1 above.)	
Image Overlays Only		
Filename	This is the location and file name of the image you want overlaid onto the selected window.	.bmp .jpg .png .pnp
Widget (both Text and Image Overlays)		
Visible	When enabled, shows the selected overlay in the window. When cleared, it hides the overlay.	Turns on or off
Interact	When enabled, allows you to move and resize the selected overlay. When cleared, it locks the size and position.	Turns on or off
Keep aspect ratio	When enabled, locks the aspect ratio of the sizing box surrounding the overlay so that when you resize it, the shape of the box is maintained. When cleared, you may change the shape of the sizing box surrounding the overlay. **Note:** The shape of the sizing box has no real effect upon the display of the overlay. No matter the shape,	Turns on or off

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

Setting	Description	Range
	Rocky will automatically maximize the size of the content to fill it.	
Position | X	For the selected overlay, this defines the X-position of the sizing box's lower left corner normalized in relation to the window size. A value of zero (0) is equal to the left window border and a value of 1 is equal to the right window border.	0-1
Position | Y	For the selected overlay, this defines the Y-position of the sizing box's lower left corner normalized in relation to the window size. A value of zero (0) is equal to the lower window border and a value of 1 is equal to the upper window border.	0-1
Position | Width	For the selected overlay, this defines the width of the sizing box normalized in relation to the window size. A value of 1 is equal to the full width of the window.	0-1
Position | Height	For the selected overlay, this defines the height of the sizing box normalized in relation to the window size. A value of 1 is equal to the full height of the window.	0-1
Opacity	Enables you to adjust the transparency of the overlay. The higher the value, the more opaque the overlay appears. The lower the value, the more transparent.	0.0 - 1.0 (Values limited by UI)
Border Size	Enables you to increase the thickness of the sizing box that surrounds the overlay.	Whole positive values including 0 (zero)
Border Color	Enables you to change the color of the sizing box that surrounds the overlay.	Options limited by the choices in the **Select Color** dialog.

What would you like to do?

- Change Window Display and Data Update Options for the Selected Particles Details Window (p. 694)
- Add or Edit a Text Overlay for a Particles Details Window (p. 692)
- Add or Edit an Image Overlay for a Particles Details Window (p. 692)
- Learn more About Exporting Data and Images Out of Rocky (p. 1155)

See Also:

- Preview a Particle Shape in 3D (p. 681)
- About Adding and Editing Particle Sets (p. 602)
- About Using the Window Editors Panel to Change the Window Axes Displays (p. 1071)

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

ADD OR EDIT AN IMAGE OVERLAY FOR A PARTICLES DETAILS WINDOW

1. Ensure the **Window Editors** panel is visible. (From the **View** menu, click **Window Editors**.)
2. From the Workspace (p. 1396), select the Particles Details window that you want to change. (See also Preview a Particle Shape in 3D (p. 681).)
3. From the **Window Editors** panel, select the **Overlays** tab and then do one of the following:
   - To add a new image, do all of the following:
     a. Click the **Add image overlay** button. A new Image entry appears in the Overlays list.
     b. Under **Input** click the **...** button next to **Filename**.
     c. From the **Load File** dialog, navigate to and select the image file you want, and then click **Open**.

   The location of the file you selected is displayed in the **Filename** box and the Image entry is appended with the file name. The image you selected appears in the selected window.

   - To edit an existing image, under **Input**, select the entry representing the image that you want to change.
4. Do any or all of the following: - To move the image, use your mouse to click the center of the image and drag it to a new location on the window. - To change the size of the image, click the image border in the window, and then drag to resize as desired.

   Note: While the **Keep aspect ratio** (under **Widget**) setting will allow you to change the shape of the sizing box that surrounds the image, it will not affect the size of the image itself. No matter the shape of the sizing box, Rocky will maximize the image size to fill it.

   - To hide the image from the window, under **Widget**, clear the **Visible** checkbox.
   - To lock the size and position of the image, under **Widget**, clear the **Interact** checkbox.
   - To change how transparent the image is, under **Widget**, edit the **Opacity** value. Tip:Click away from the Opacity box to have your changes applied to the window.
   - To change the appearance of the box bordering the image, choose what you want under **Widget** for **Border Size** and **Border Color**.
   - To remove the image from the selected window, click the **Remove overlay** button.

See Also:

- About Using the Window Editors Panel to Change the Selected Particles Details Window (p. 683)
- Change Window Display and Data Update Options for the Selected Particles Details Window (p. 694)
- Add or Edit a Text Overlay for a Particles Details Window (p. 692)

ADD OR EDIT A TEXT OVERLAY FOR A PARTICLES DETAILS WINDOW

1. Ensure the **Window Editors** panel is visible. (From the **View** menu, click **Window Editors**.)

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

2. From the Workspace (p. 1396), select the Particles Details window that you want to change. (See also Preview a Particle Shape in 3D (p. 681).)

3. From the **Window Editors** panel, select the **Overlays** tab and then do one of the following:

   - To edit an existing text overlay, select the name of the text you want to change from the list at the top of the panel.

   - To add a new text overlay, click the **Add text overlay** button. A new Text entry appears in the Overlays list at the top of the panel.

4. Do any or all of the following:

   - To enter your own text, under **Input**, place your cursor directly into the Text entry and then remove and add text as you like using your keyboard.

   - To have Rocky automatically display simulation-related text, including simulation time or study name, in the **Input** box, place your cursor where you want the text to be inserted, choose what you want from the **Keyword for** list, and then click **Insert**. The placeholder text appears in the **Input** box. Tip:To see the text replaced with real values on the selected window, click **Apply**.

   - To change how the text is displayed in the window, edit the **Bold**, **Italic**, **Scale Mode**, **Font Size**, **Justification**, and **Color** options as you want. (See also About Using the Window Editors Panel to Change the Selected Particles Details Window (p. 683).)

   - To move the text, do one of the following:

     – Use your mouse to click the center of the text box and then drag it to a new location on the window.

     – Under **Widget**, use the **Position** options for **X** and **Y**.

   - To change the size and shape of the text box, do one of the following:

     – Click the text box border in the window, and then drag the border to resize or reshape the text box.

     – Under **Widget**, use the **Position** options for **Width** and **Height**.

     Note:

     While the **Keep aspect ratio** (under **Widget**) setting will allow you to change the shape of the sizing box that surrounds the text, it will not affect the size of the text itself. No matter the shape of the sizing box, Rocky will maximize the text size to fill it.

   - To hide the text from the window, under **Widget**, clear the **Visible** checkbox.

   - To lock the size and position of the text box, under **Widget**, clear the **Interact** checkbox.

   - To change how transparent the text is, under **Widget**, edit the **Opacity** value. Tip:Click away from the **Opacity** box to have your changes applied to the selected window.

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

- To change the appearance of the box surrounding the text, choose what you want under **Widget** for **Border Size** and **Border Color**.
- To remove the selected overlay from the window entirely, click the **Remove overlay** button.

See Also:

- About Using the Window Editors Panel to Change the Selected Particles Details Window (p. 683)
- Change Window Display and Data Update Options for the Selected Particles Details Window (p. 694)
- Add or Edit an Image Overlay for a Particles Details Window (p. 692)

CHANGE WINDOW DISPLAY AND DATA UPDATE OPTIONS FOR THE SELECTED PARTICLES DETAILS WINDOW

1. Ensure the **Window Editors** panel is visible. (From the **View** menu, click **Window Editors**.)
2. From the Workspace (p. 1396), select the Particles Details window that you want to change. (See also Preview a Particle Shape in 3D (p. 681).)
3. From the **Window Editors** panel, select the **Particles Details View** tab and then change the options you want.

Tips:

- You can also open the **Window Editors** panel for a window by right-clicking an empty space within the Particles Details window (for example, the background behind the particle), and then clicking **Settings**.
- You can also use Rocky Preferences to change the background color and fonts for all the Particles Details windows in your project, not just selected one. (See the "Preferences for Windows Properties" section in the About Setting Global Preferences (p. 227) topic.)
- To reuse in future projects the settings you made to this tab, click the **Save Current Configuration in Settings** button.
- To apply settings you have already saved (either by previously using the **Save Current Configuration in Settings** button or by saving selections within the **Preferences** dialog) to this tab, click the **Restore Configuration from Settings** button.

See Also:

- About Using the Window Editors panel to Change the Selected Particles Details Window (p. 683)
- About Using the Window Editors Panel to Change the Window Axes Displays (p. 1071)
- Add or Edit a Text Overlay for a 3D View or Particles Details Window (p. 1069)
- Add or Edit an Image Overlay for a 3D View or Particles Details Window (p. 1070)

USING THE COLORING TAB TO CHANGE A PARTICLES DETAILS WINDOW

The Coloring tab, located on the Data Editors panel for all simulation entities and User Processes, enables you to change the colors and data attributes of the particle shape being displayed in a

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

Particles Details window. Faces, edges, and nodes can be shown or hidden; displayed in solid colors; or colored by a Property property of your choice.

In addition, when showing the results of Collisions Statistics, the Coloring tab also enables you to modify the limits, units, and legend options for the color scales used in your Particles Details windows.

What would you like to do?

- Learn more About Using the Coloring Tab to Change the Preview for a Particle Set (p. 695)
- Learn more About Color Scales (p. 1086)
- Use the Coloring Tab to Change the Preview for a Particle Set (p. 697)

See Also:

- Change a Particles Details Window (p. 682)
- About Intra-Particle Collision Statistics (p. 324)

ABOUT USING THE COLORING TAB TO CHANGE THE PREVIEW FOR A PARTICLE SET

Using the options on the Coloring tab is one way you can change what is displayed in a Particles Details window for the Particle set (p. 1393) that you want to change. Visualization options for the Faces, Edges, and Nodes can be modified with the Coloring tab.

When analyzing the results of Intra-particle Collision Statistics, you may from the Coloring tab also adjust the Color scales for the Property you are viewing. However, the options presented are quite different than the ones explained below. For more information, see also About Color Scales (p. 1086).

PARAMETER DEFINITIONS

See the image and table below to help you understand how to use the Coloring tab to change a view for selected entity.

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

Figure 3.146: Example Coloring tab in the Data Editors panel showing Particle Set parameters for a Particles Details window

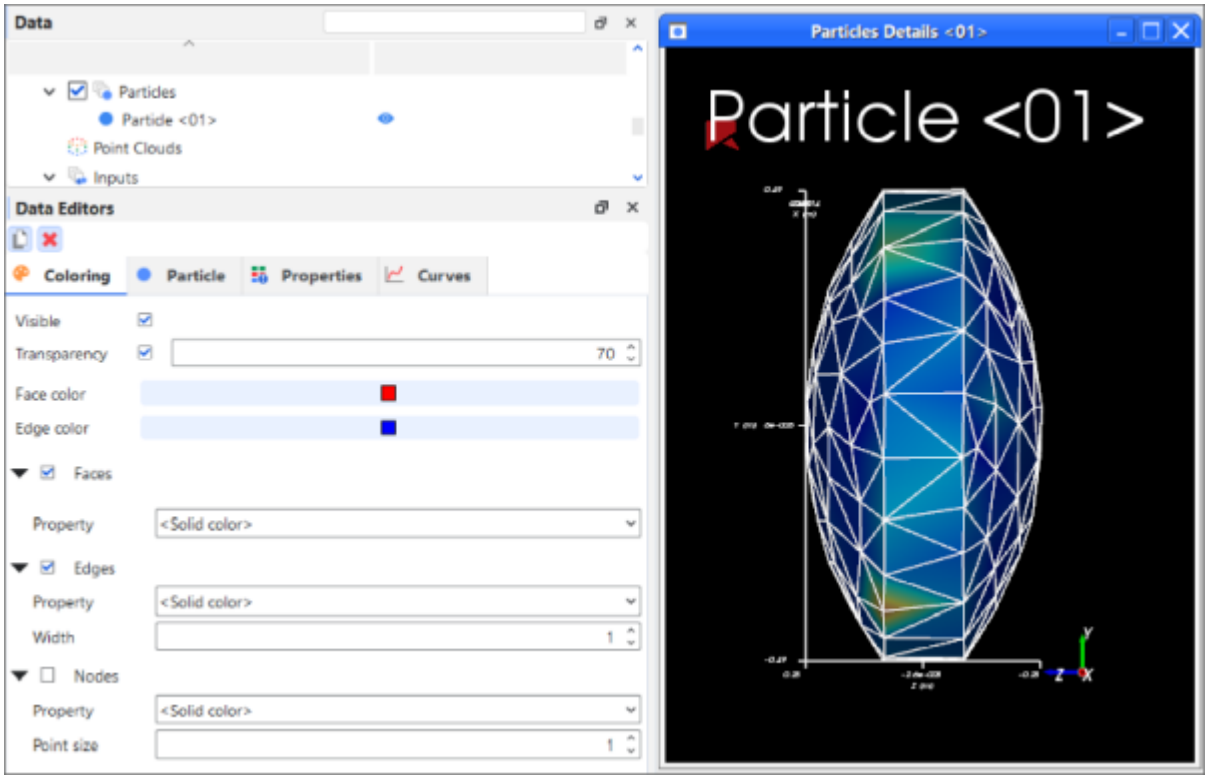


Table 1: Coloring tab parameter options for Particle sets in the Particles Details window

Setting	Description	Range
Visible	When enabled, shows the selected entity in the active view window. **Note:** This is the same as using the Data panel eye icon. (See also Show/Hide Components by Using Eye icons and Checkboxes (p. 113)).	Turns on or off
Transparency	Enables boundaries to be drawn transparently in the selected view window. A value of 0 (zero) makes the faces completely solid or opaque. A value of 100 makes the faces completely transparent or hidden.	Turns on or off; 0-100
Color	When **Faces**, **Edges**, and/or **Nodes** are selected and the **Property** is <Solid Color>, this enables the selected display type to be colored in the one solid color chosen. Note: For **Faces**, **Transparency** options affect how opaque this color appears in the view.	Options limited by the **Select Color** dialog
Faces	Enables the entire surface of the entity to be visible and affected by the various Faces coloring options.	Turns on or off

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

Setting	Description	Range
Property	Three separate lists under **Faces**, **Edges**, and **Nodes** respectively provide properties and other color options that apply to the display type within which the list is contained. Specifically: • **<Solid Color>** applies a single color to the display type. • The remaining properties come directly from the component's Properties tab. Selecting one of these properties colors display type in a range according to the data collected for the current time step. (See also About Properties (p. 933)).	<Solid color>; List of properties automatically generated from the Properties tab
Edges	Enables only the edges bordering the components making up the entity boundaries visible and affected by the various Edges coloring options.	Turns on or off
Width	When **Edges** is selected, this changes the thickness of the lines used to draw the borders. Entering 1 will provide the narrowest line.	Positive values
Nodes	Enables the points marking the intersections of the entity's individual components to be visualized as dots and affected by the various Nodes coloring options.	Turns on or off
Point Size	When **Nodes** is selected, this changes the size of the dots used to draw the nodes. Entering 1 will provide the smallest dot.	Positive values

What would you like to do?

- Use the Coloring Tab to Change the Preview for a Particle Set (p. 697)
- Learn more About Color Scales (p. 1086)

See Also:

- Change a Particles Details Window (p. 682)
- Add and Edit Particle Sets (p. 602)

USE THE COLORING TAB TO CHANGE THE PREVIEW FOR A PARTICLE SET

1. From the Workspace (p. 1396), select the Particles Details window containing the *Particle set* (p. 1393) to which you want to apply changes. (See also Preview a Particle Shape in 3D (p. 681).)
2. From the **Data** panel, under **Particles**, select the Particle set you want to change.
3. From the **Data Editors** panel, select the **Coloring** tab, and then change the settings you want. The changes you make are shown automatically in the selected Particles Details window.

See Also:

- About Using the Coloring Tab to Change the Preview for a Particle Set (p. 695)

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

- Change a Particles Details Window (p. 682)
- About Particles Details Windows (p. 679)

REMOVE AN EXISTING PARTICLE SET

- From the **Data** panel, under **Particles**, right-click the name of the Particle set you want to remove, and then click **Remove Particle**.

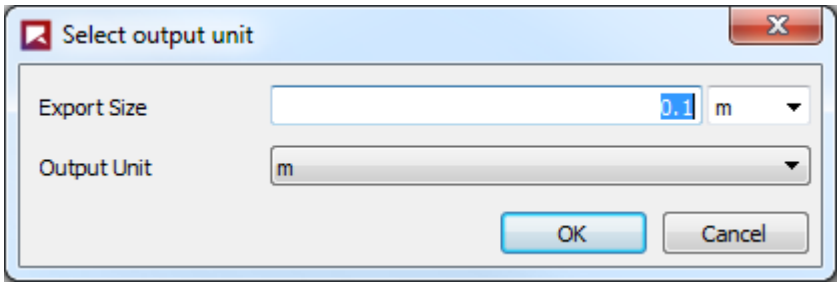
See Also:

- About Adding and Editing Particle Sets (p. 602)
- Add a New Particle Set (p. 678)
- Modify the Parameters of a Particle Set (p. 678)

EXPORT A PARTICLE SHAPE TO AN STL FILE

In cases where you need to make use of your particle shape outside of Rocky, you may choose to export your rendered shape (p. 1394) (any particle shape) or your originally imported shape (Custom Particle Shapes only) to an .stl file. Doing so enables you to select the size and units you want to use when exporting (Figure 1).

Figure 3.147: Select Output Unit dialog



To export a particle shape to an STL file:

1. From the **Data** panel, under **Particles**, right-click the Particle set containing the shape you want to export, point to **Export**, and then do one of the following:
   - For default and custom imported particle shapes, click **Rendered Shape** to export Rocky's current (and perhaps altered) copy of the shape file.
   - For custom imported particle shapes only, click **Imported Shape** to export the original (unaltered) file.
2. From the **Select output unit** dialog, enter the **Export Size**, select from the **Output Unit** list the units you want, and then click **OK**.
3. From the **Select target STL file** dialog, click the drive or folder of the location to which you want to save the file.

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

4. In the **File name** box, enter a name for the file, and then click **Save**.

Note:

If you are wanting to export a rendered (p. 1394) Shell (p. 1395) shape out of Rocky, be aware that the exported Shell shape will no longer be considered a true Shell (2D) shape due to the thickness Rocky applies to its walls. It is for this reason that it is recommended that you do not re-import a previously exported rendered Shell shape back into the Rocky program.

See Also:

- About Adding and Editing Particle Sets (p. 602)
- About Defining and Importing Custom Particle Shapes (p. 670)
- About Exporting Data and Images Out of Rocky (p. 1155)

3.2.13. Point Clouds

Point Clouds enable you to import a text file containing field data that defines one or more points in space.

Tip:

Unless you have enabled an external Module (p. 1391) that makes use of Point Clouds, you can skip the Point Clouds part of your Rocky setup.

What would you like to do?

- Learn more About Point Clouds (p. 699)
- Add a Point Cloud (p. 704)
- Edit or Remove a Point Cloud (p. 705)

See Also:

- About Modules Parameters (p. 431)
- Set Simulation Parameters (p. 414)

ABOUT POINT CLOUDS

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

In this version of Rocky, the primary purpose of Point Clouds are to support the functionality within custom external Modules (p. 1391) whose models require information that varies through all three dimensions of the domain and/or over time.

Tip:

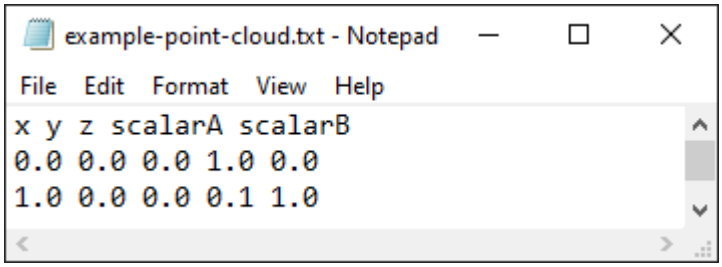
Unless you have enabled an external Module that makes use of Point Clouds, you can skip the Point Clouds part of your Rocky setup.

Specifically, Point Clouds enable you to import a text, csv or xlsx file containing field data that defines one or more static points in space. Moreover, you can load a set of point clouds at once by importing a transient point cloud text file, which specifies the start and end times for individual static point clouds. This data can then be used by Rocky in various ways depending upon the functionality of the external Module (p. 1391) making use of the Point Cloud.

(See also About Modules Parameters (p. 431).)

About Creating a Static Point Cloud Definition File

Figure 3.148: Example text file defining a Static Point Cloud with two variables and two points



The text file you import for a Static Point Cloud must be formatted in the following way:

- The file must be saved as a .txt extension with spaces separating each value. Important: No matter your system's location settings, ensure that you use dots (periods) and do not use commas to indicate decimals in this particular text file.
- The first (header) row must contain the names of each value defined.
- The first three (required) columns must be `(x, y, z)`, which together define the coordinate location of the point.
- The fourth (required) column must include a scalar variable. (The variable you include will be dependent upon how the Point Cloud will be used.)
- The fifth and subsequent columns (optional) can include additional scalar variables as needed.
- The second and subsequent rows define the values for each individual point.

Rocky allows you to create Static Point Clouds with different dimensions and units. The column name now can carry the dimension of a point cloud property. For the coordinates the default unit is **[m]**, for properties user can explicitly define a set of valid dimensions after the property name like:

- Velocity [m/s]

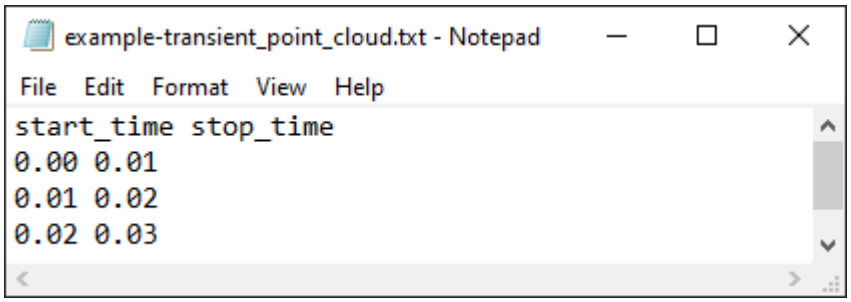
Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

- Force [N]
- Temperature [K]
- Pressure [Pa]

Columns with the missing "**[dim]**" dimension will throw a warning about having a dimensionless property if not configured later in a module setup.

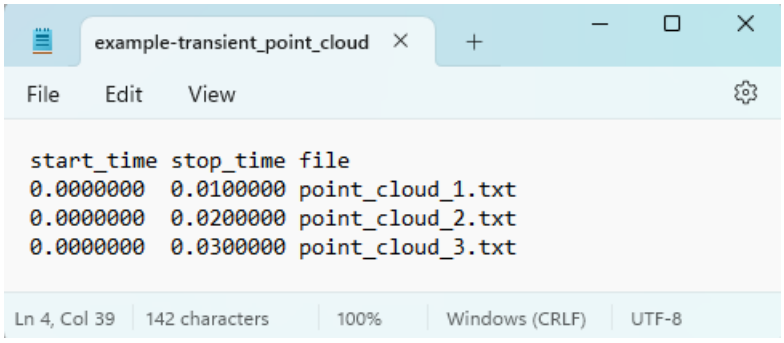
About Creating a Transient Point Cloud Definition File

Figure 3.149: Example text file defining the main file of a Transient Point Cloud with three Static Point Clouds.



Transient Point Clouds consist of sets of Static Point Clouds that are applied during the simulation following specific start and end times. In order to import a Transient Point Cloud, it is necessary to import a main text, csv or xlsx file containing the start and end times of each Static Point Cloud that will be then automatically imported to compose the Transient Point Cloud. If desired, an optional third column with the name of the files (with the file extension) for each start and stop time, as shown below:

Figure 3.150: Example text file defining the main file of a Transient Point Cloud with three Static Point Clouds.



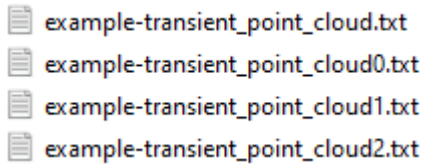
The main file you import for a Transient Point Cloud must be formatted in the following way (see Example text file defining the main file of a Transient Point Cloud with three Point Clouds (p. 701)):

- The file must be saved as a .txt, .csv or .xlsx extension with spaces separating each value. Important: No matter your system's location settings, ensure that you use dots (periods) and not commas to indicate decimals in this particular text file.
- The first (header) row must contain only the **start_time** and the **stop_time** names and, optionally, the **file** names.

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

- The second and subsequent rows define the start time and the stop time values for the Static Point Clouds that you want to be imported. If you decided to use the file column, it also needs to be filled here.
- The start time value in the first row not necessarily needs to be zero.
- The stop time entry in the last row can be empty so the last Static Point Cloud will be enabled until the end of the simulation.
- All rows must have a file set in the file column if you decided to use this feature.

Figure 3.151: Example text files structure to define a Transient Static Point Cloud with three Static Point Clouds.



The Static Point Clouds files that makes up the Transient Point Clouds must have the same format as the regular Static Point Clouds. (See also About Creating a Static Point Cloud Definition File (p. 700).)

The Transient Point Cloud files structure must follow these rules (see Example text file defining the main file of a Transient Point Cloud with three Point Clouds (p. 702)):

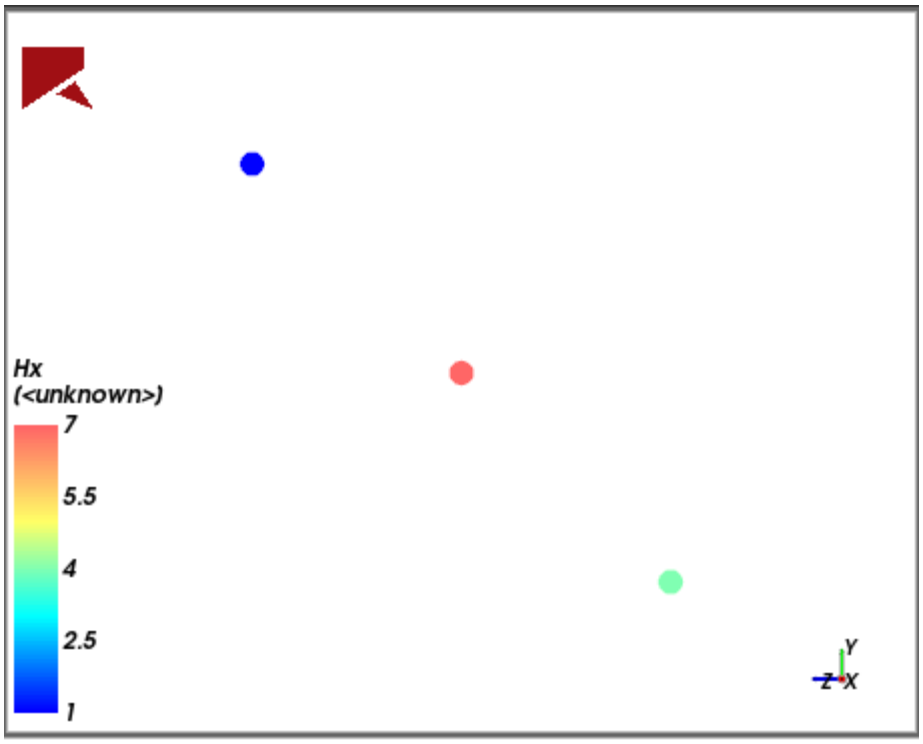
- The main Transient Point Cloud file and each Static Point Cloud files must be located at the same folder.
- The filename of each of the Static Point Clouds must be the same one used for the main Transient Point Cloud txt, csv or xlsx file, but adding an index at the end of the filename. If you decide to use the file column, you may create any names for the Static Point Clouds, but they need to be specified in the File column in each row.
- The start and the stop times defined in the main file rows will be considered to load the Static Point Clouds into the simulation. Each row is related to each Static Point Cloud index, starting from 0.

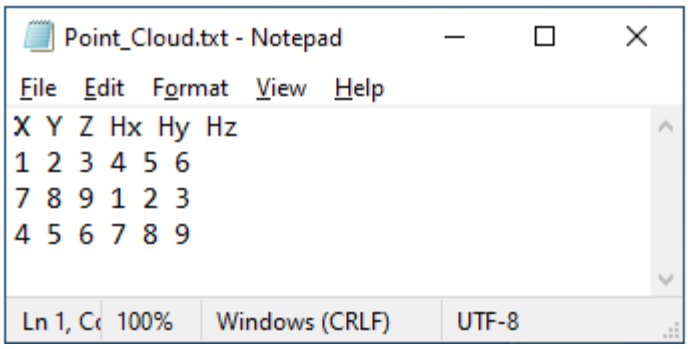
Visualizing a Point Cloud

In this version of Rocky, you can visualize in a 3D View window the individual points defined within your Point Cloud Definition File (Figure 2).

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

Figure 3.152: A Point Cloud being visualized in a 3D View window





(See also About Using the Coloring Tab to Change a 3D View (p. 1074).)

Point Cloud Parameter Definitions

Use the image and table below to help you understand the settings for a Point Cloud.

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

Figure 3.153: Point Cloud parameters in the Data Editors panel

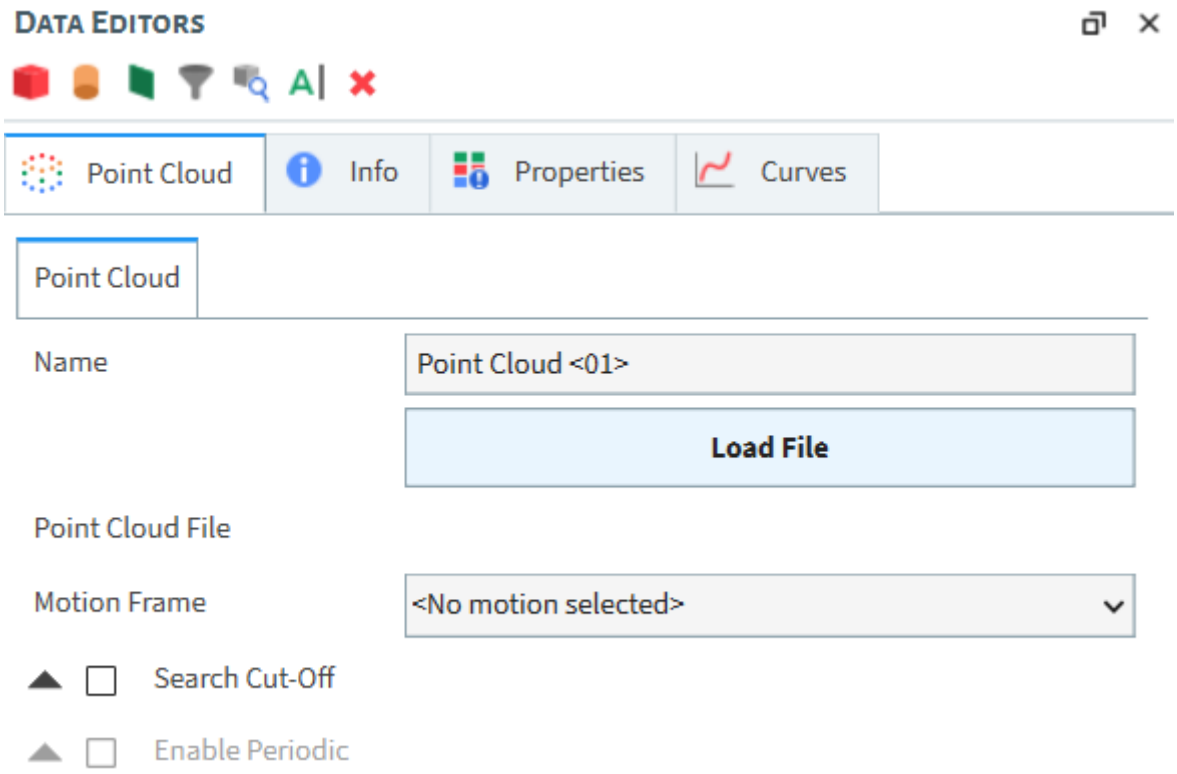


Table 1: Point Cloud parameter options

Setting	Description	Range
Name	Enables you to specify a unique identifier for the Point Cloud.	99 character limit
Motion Frame	Applies motion frames to the Point Cloud.	—
Search Cut-Off	Skips nearest search beyond a set distance to improve performance.	On / Off + Distance (m)

What would you like to do?

- Add a Point Cloud (p. 704)
- Edit or Remove a Point Cloud (p. 705)

See Also:

- About Modules Parameters (p. 431)
- About Using the Coloring Tab to Change a 3D View (p. 1074)
- View Geometries, Particles, Points, and Fluids in 3D (p. 1041)

ADD A POINT CLOUD

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

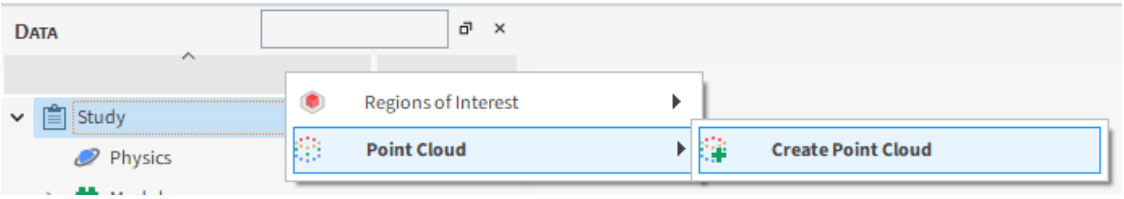
1. Ensure you have enabled an external Module (p. 1391) that makes use of Point Clouds.

> **Tip:**
>
> If you do not plan to enable an external Module that makes use of Point Clouds, then you do not need to add a Point Cloud to your simulation.

2. Ensure you have a Point Cloud definition text file that is formatted according to the way Rocky expects. (See also About Point Clouds (p. 699).)

3. From the **Data** panel, right-click **Study**, select **Point Cloud**, and then click **Create Point Cloud**.

Figure 3.154: How to create a Point Cloud



4. From the **Data** panel, under **Point Clouds** select the newly added Point Cloud entry.

> **Note:**
>
> The **Point Clouds** entity shows up only if you create a Point Cloud, otherwise it will be hidden from the Study.

5. From the **Data Editors** panel, on the main **Point Cloud** tab, enter a **Name** and then click the **Load File** button.

6. From the **Select file to import** dialog, navigate to and select the .txt file you want to use, and then click **Open**.

See Also:

- About Point Clouds (p. 699)
- About Modules Parameters (p. 431)

EDIT OR REMOVE A POINT CLOUD

1. From the **Data** panel, under **Point Clouds** select the Point Cloud entry you want to change.

2. From the **Data Editors** panel, do one of the following:

   - To edit the Point Cloud, from the main **Point Cloud** tab, do any of the following:

     – Enter a different **Name**.

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

  - Click the **Load File** button and then from the **Select file to import** dialog, navigate to and select the new .txt file you want to use, and then click **Open**.

- To remove the Point Cloud, click the **Remove Point Cloud** button.

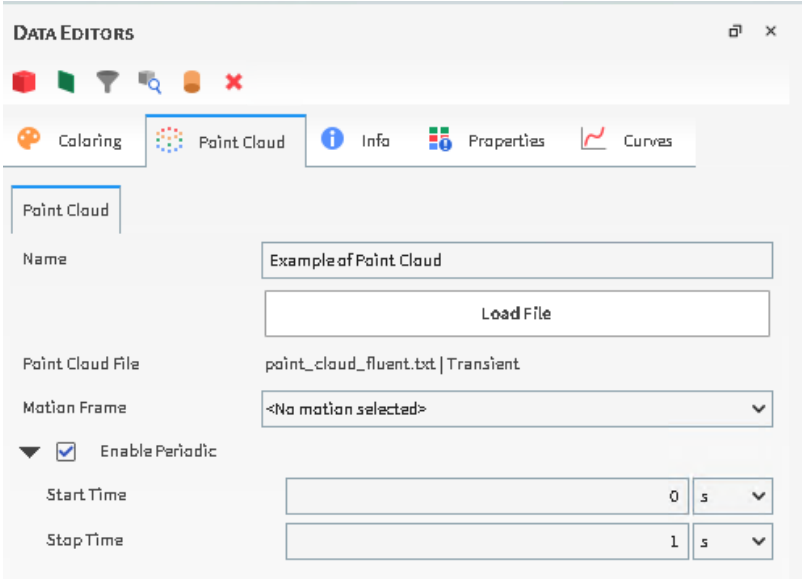
TIME PERIODIC FOR TRANSIENT POINT CLOUDS

1. For the latest Rocky release, the support for Time Periodic was added to the Transient Point Clouds capabilities, enabling to execution of the whole Point Cloud data as periodic, **repeating** its data respecting the Start and Stop time that is available to configure in the **Data Editors**.

2. To edit the Time Periodic for the Transient Point Cloud, from the main **Point Cloud** tab, do the following

   - Check the Enable Periodic option in the **Data Editors**, and then, enter a **Start Time** and a **Stop Time**, which will control the times that the Periodic will begin and be finished. Then, the Point Cloud Data will repeat, constantly, respecting the Start-Stop time which were chosen.

   - **Figure 3.155: How to set the Time Periodic for the Point Cloud**



MOTION FRAME SUPPORT TO POINT CLOUD

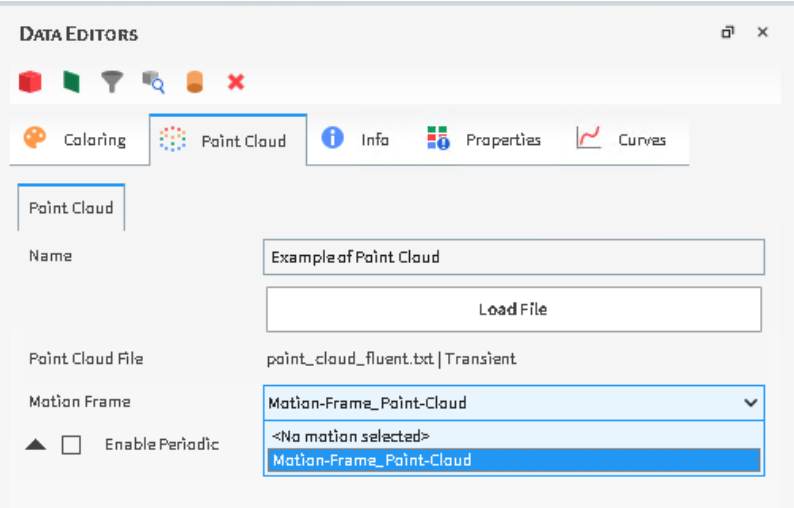
1. For the latest Rocky release, the support for Motion Frame was also added to Point Cloud capabilities, enabling to move of the Point Cloud data accordingly with the already configured **Motion Frame**, that can be selected in the **Data Editors**. This Motion Frame support is available for both Static and Transient Point Cloud data, as well it works with the new **Time Periodic** supprt.

2. To edit the Motion Frame for the Point Cloud, from the main **Point Cloud** tab, do the following

   - After the Motion Frame has already been created, select it in the **Motion Frame** selection box, which will shown all the Motion Frames availabe in the simulation, as shown in the image below.

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

- **Figure 3.156: How to set the Motion Frame for the Point Cloud**

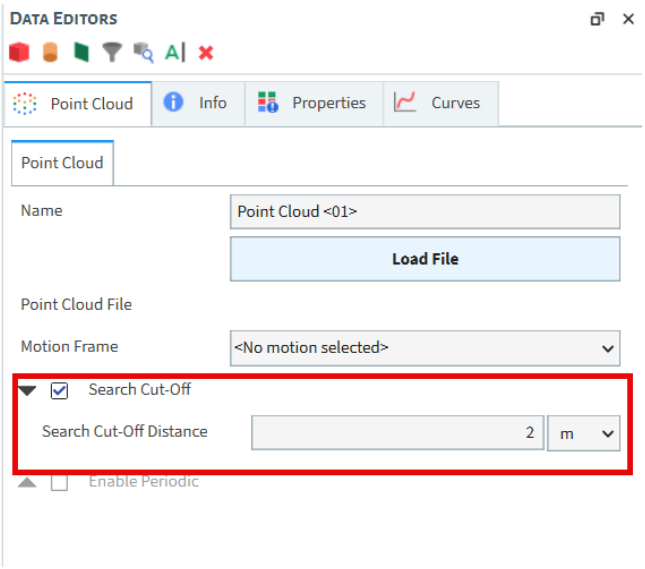


SEARCH CUT-OFF

The **Search Cut-Off** feature improves performance during nearest-point searches in point clouds. When enabled, the solver skips searching for neighbors that are farther than a specified physical distance, reducing computation time.

To use it, enable **Search Cut-Off** and set the **Distance** parameter to define the region where particles will be ignored. Points outside this distance will not have a valid neighbor index. Use larger distances for detailed searches and smaller ones for faster performance.

Figure 3.157: Search Cut -Off options for the Point Cloud



See Also:

- About Point Clouds (p. 699)
- About Modules Parameters (p. 431)

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

- Add a Point Cloud (p. 704)

ADD AND EDIT INLETS AND OUTLETS

The **Inlet and Outlets** entity on the Data panel is where you specify how and where various particle and fluid sets will be entering (or leaving) your simulation-for example, through continuous injection, volumetric inlet, or custom positioning that you define in a text or spreadsheet file-and get an estimate of how many particles might be released from those inlet locations during the simulation.

What would you like to do?

- Learn more About Adding and Editing Inlets and Outlets (p. 708)
- Create a New Particle Inlet (p. 740)
- Create a New Fluid Inlet (p. 741)
- Create a New Volumetric Inlet (p. 741)
- Create a New Custom Input for Particles (p. 743)
- Edit an Inlet or an Outlet (p. 745)
- Remove an Inlet or an Outlet (p. 745)

See Also:

- Set Simulation Parameters (p. 414)

3.2.14. Adding and Editing Inlets and Outlets

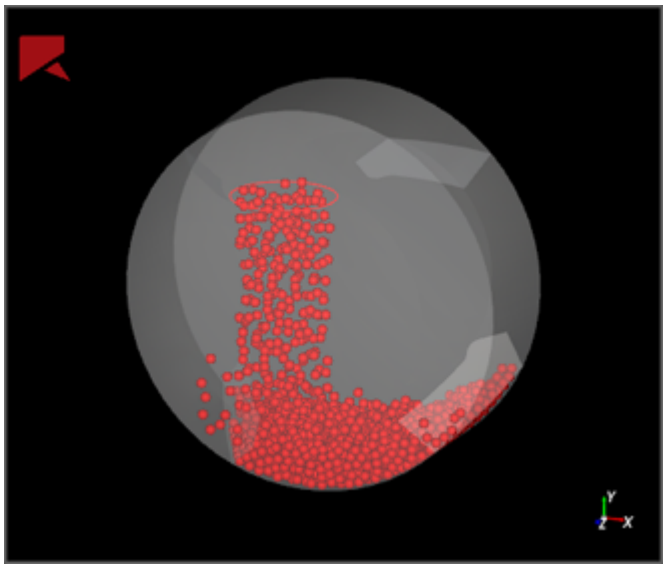
Inlets define how particles or fluids are injected into a simulation. Outlets define the surface(s) where particles or fluids leave the simulation. In this version of Rocky, there are four different inlet methods (Particle Inlet, Particle Custom Inlet, Volumetric Inlet, and Fluid Inlet) and one outlet option. See the sections below to learn more about adding and editing the various kinds of inlets and outlet in Rocky.

PARTICLE INLET

The **Particle Inlet** method enables you to release particles in a continuous or periodic stream from the **Entry Point** (Surface or Feed Conveyor) that you select (Figure 1).

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

Figure 3.158: Example of Particle Inlet from a Circular Surface



With this method, the size and shape of both the **Entry Point** and the particles within the set (see also About Adding and Editing Particle Sets (p. 602)), as well as the **Mass Flow Rate** and particle overlap criteria that that you define (see also About Solver Parameters (p. 464)) all have an effect on the injection speed and relative placement (or "packing") of the particles as they are released.

In addition, you are able to delay or halt particle injection at various times in the simulation.

FLUID INLET

The **Fluid Inlet** method enables you to set an inlet of fluid into your simulation. With this method, the temperature of the fluid (if a thermal model is enabled) and the surface that will serve as inlet can be defined. Additionally, you can choose to set the **Mass Flow Rate** or the **Velocity** as a boundary condition and be able to delay or halt fluid injection at various times in the simulation.

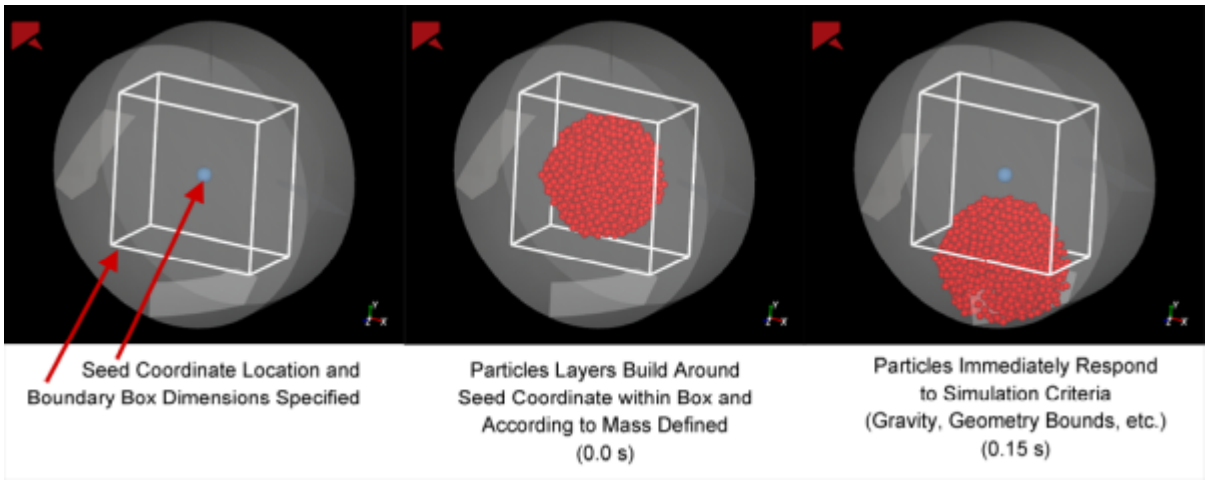
VOLUMETRIC INLET

The **Volumetric Inlet** method enables you to inject a spheric region of closely packed particles into a simulation all at one time (Figure 2). Alternatively you can use this method to create a region that will be filled with fluid, using the SPH tab.

Volumetric Inlet for Spherical Particles

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

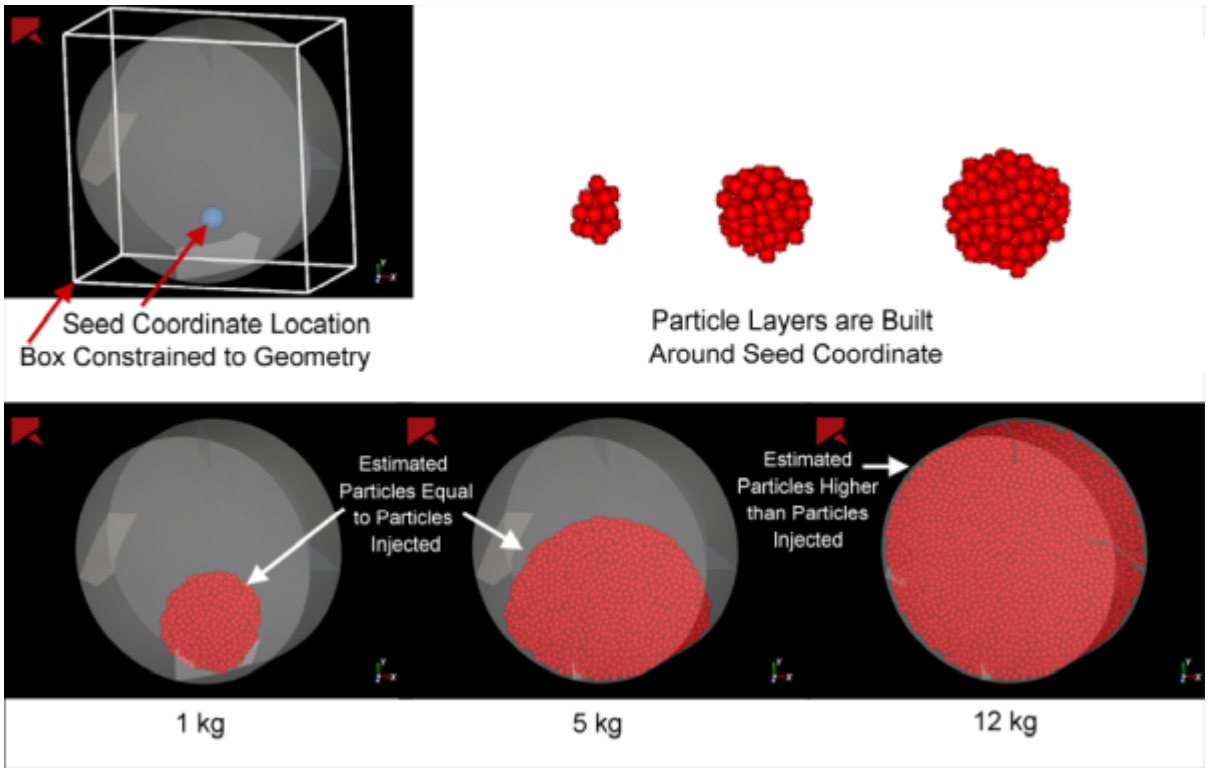
Figure 3.159: Example of a Particle Volumetric Inlet



Rocky does this by building layers of particles around a central **Seed Coordinate** point you specify. These layers continue to build making the sphere bigger until one of two things happens: it either meets the **Mass** value that you set or it reaches the limits you define-whichever comes first. The limits you define must include a boundary box-the volume of which Rocky uses to limit the particle fill- but can also include the walls or surfaces of one or more boundaries within your simulation.

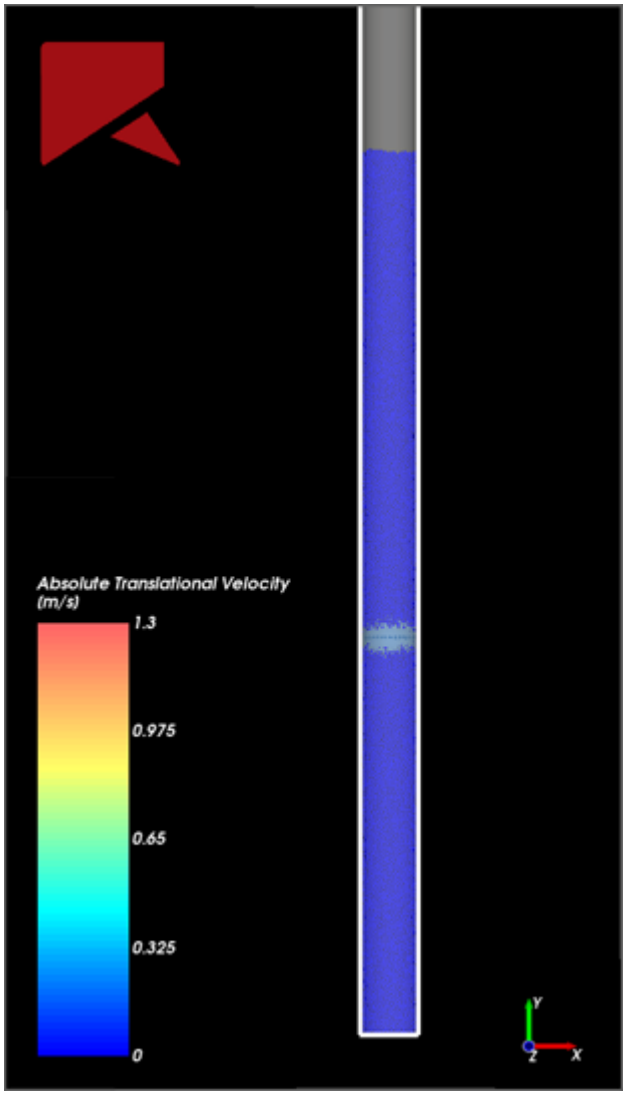
Choosing to constrain the fill by a geometry (wall or surface) enables you to have the fill build within, around, or through objects, which can be useful in certain situations (Figures 3-4).

Figure 3.160: Example of a Volumetric Inlet being constrained by a drum geometry (wall or surface)



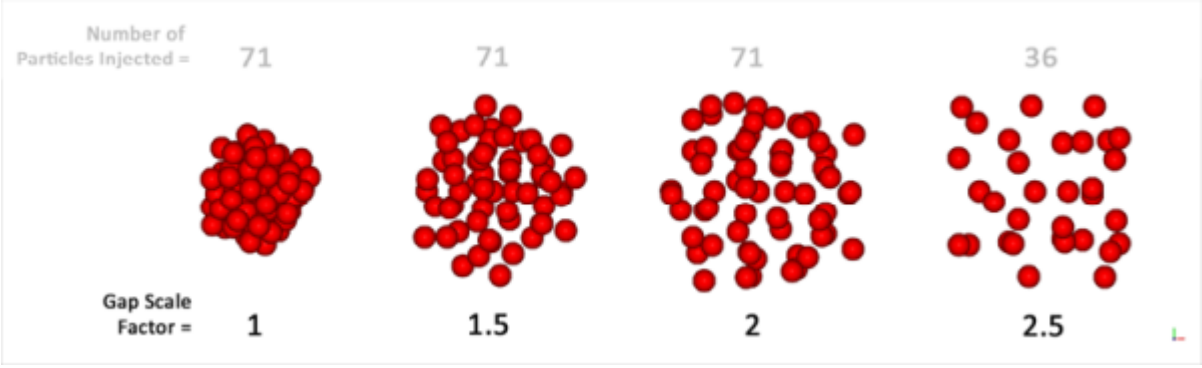
Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information
of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

Figure 3.161: Example of a Volumetric Inlet following along the walls of the pipe geometry to which the inlet was constrained



How closely the particles are to each other when they build around the Seed Coordinate (the "gap" between them) is determined by the **Gap Scale Factor** you set. A default factor of 1 places the particles right next to each other; a factor of 2 places the particles as if they were twice the size, and so on (Figure 5).

Figure 3.162: Comparison of different Gap Scale Factors in a Volumetric Inlet



Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information
of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

Defining a factor that is too big for your **Box bounds** will result in fewer particles being injected-as shown in the last example in Figure 5.

Note: Volumetric Inlets require a minimum of 3 particles in order to create the "Seed". You may need to adjust your **Mass** and other Volumetric Inlet properties to ensure that Rocky can generate enough particles to meet this requirement.

With the Volumetric Inlet method, the size and shape of the particles within the set (see also About Adding and Editing Particle Sets (p. 602)) and the set's Material properties (see also About Modifying Material Compositions (p. 591))-not to mention the limits and **Gap Scale Factor** you define-all work together to determine how the **Mass** you enter translates into the amount of particles to be injected.

While the Volumetric Inlet option serves to provide maximum flexibility with regards to your set up, this complexity can also increase your chances for mistakes. Please review My Volumetric Inlet isn't working right (p. 1360) to learn how to avoid the most common pitfalls.

Tip:

To see walk-through examples using the Volumetric Inlet, refer to the following Workshops:

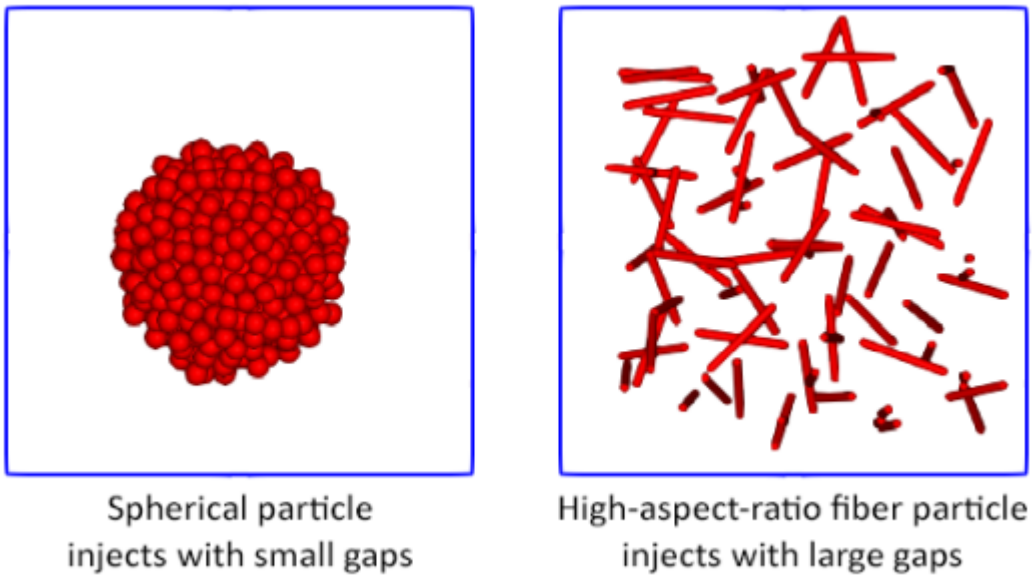
- Workshop 04: SAG Mill
- Workshop 07: Conical Dryer
- Workshop 09: Tablet Coater
- Workshop 14: DEM-CFD Two-Way Coupling with Ansys Fluent
- Workshop 18: Mixing Analysis (Liquid Bridge and LMI)
- Workshop 19: Backhoe Loader
- Tutorial - Dam Break in the *Rocky Tutorial Guide*
- Tutorial - Slurry Mill in the *Rocky Tutorial Guide*

Volumetric Inlet for Non-spherical Particles

Volumetric Inlet for non-spherical shapes considers the biggest distance between two points—i.e., the longest length within the boundaries of the particle—and uses that distance to create a spherical buffer around the particle that exists only during the initial time step of the injection. This buffer prevents other particles—no matter their orientation—from going inside another particle's buffer. While this strategy prevents overlaps during injection, it can also cause wider "gaps" between some types of particles. As the biggest length of the non-spherical shape increases, the distance between particles (also known as the "gap" between them) increases, and fewer particles are injected during Volumetric Inlet.

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

Figure 3.163: Comparison between spherical and non-spherical particles in a Volume-fill



In the illustration shown in Figure 6, spherical particles and high-aspect-ratio fiber particles of the same initial mass and tonnage are injected via separate but identical Volumetric Inlet regions. While the spherical particles are injected closely together, the fiber particles are injected much farther apart due to the spherical buffer required for overlap control. And as a result, the fiber particles reach the limits of the Volumetric Inlet region much sooner, thereby injecting fewer particles.

Tip:

For more information about how overlaps are calculated in Rocky, refer to Figure 2.1 in the Rocky DEM Technical Manual.

Volumetric Inlet for Fluids

The Volumetric Inlet for fluids option (available through the SPH tab inside Volumetric Inlet), allows for the injection of fluid in a controlled region. It works in the same fashion as the particle volumetric inlet, including the possibility of using one or more geometries to define the region that will be filled with fluid.

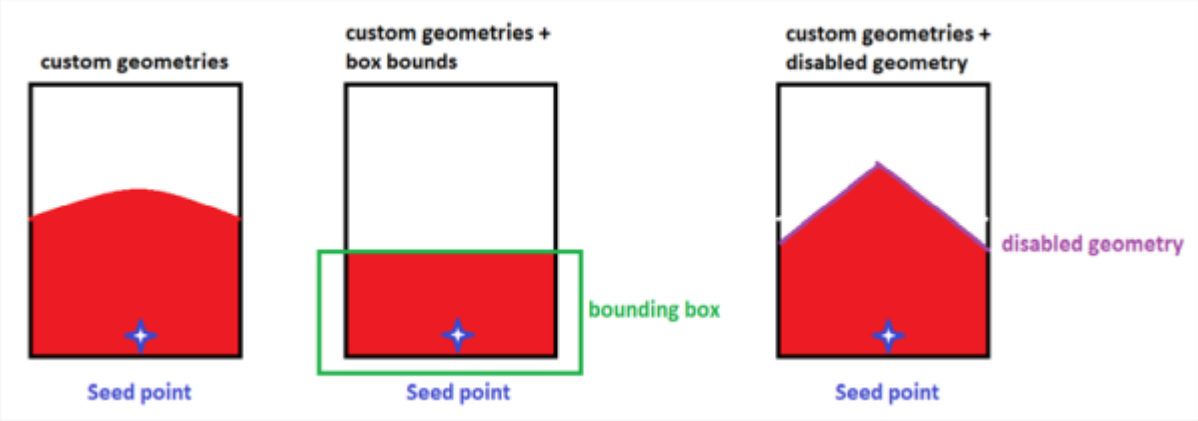
The main difference between the fluid and particle volumetric inlet is that the fluid input parameters are only the fluid mass and temperature (available only if a thermal model is enabled).

Tips for Using Volumetric Inlet

If desired, you can combine the usage of the imported geometries and the bounding box to achieve flat surfaces.

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

Figure 3.164: Different methods of limiting the Volumetric Inlet



In the first image in the example above, you can see the particle bed that is generated when the walls are used as the imported geometry for the Volumetric Inlet. In the second image, you can see what happens if you use the same imported geometry but add a bounding box that limits the particle generation. As the particles reach the bounding box, Rocky cannot inject more particles, and the surface of the Volumetric Inlet is flat.

If you want to limit the inlet to a surface that is not flat, you can use the following workaround:

- Import a geometry representing the profile of your desired top surface-such as the peak used in the third image in the example above.
- Set this geometry to be used only the initialization period-which is when the volume fill happens. You can accomplish this by setting the geometries **Enable Time** to be a very large value that ever be reached in the simulation, such as 999 s.

Particle Custom Inlet

The **Particle Custom Inlet** method enables you to position particles in the exact locations, times, amounts, sizes, speeds, and orientations that you want. You accomplish this by specifying certain values in a special definition file that you import into Rocky, and then assign a Particle set (p. 1393) to it.

ABOUT CREATING A PARTICLE CUSTOM INLET DEFINITION FILE

The definition file you create for the Particle Custom Inlet determines when and how particles are placed inside your domain. The resulting spreadsheet can be saved in various formats, such as .csv (Figure 8), .xls, .xlsx (Figure 9), .xlsm, .xlsb, and .odf. (See also File Types and Folders in Rocky (p. 129).)

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

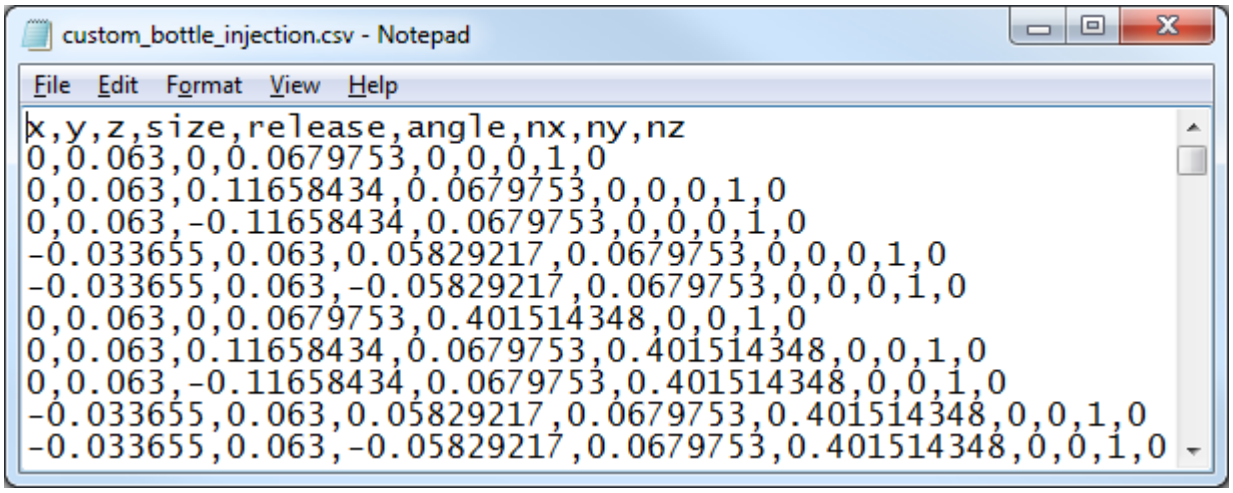
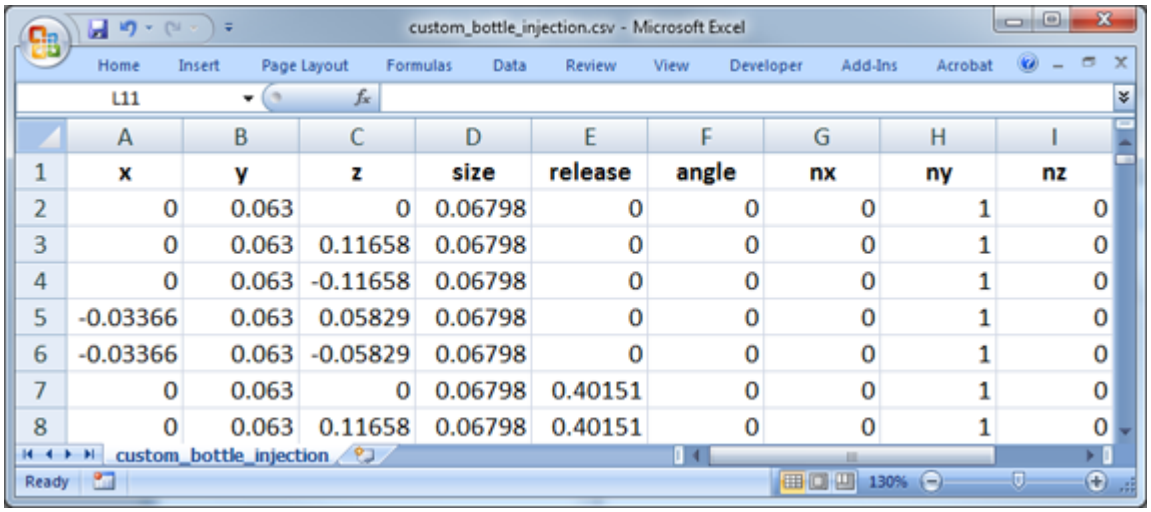


Figure 8: Example of a Particle Custom Inlet definition file showing a .csv file with commas separating the columns



	A	B	C	D	E	F	G	H	I
1	x	y	z	size	release	angle	nx	ny	nz
2	0	0.063	0	0.06798	0	0	0	1	0
3	0	0.063	0.11658	0.06798	0	0	0	1	0
4	0	0.063	-0.11658	0.06798	0	0	0	1	0
5	-0.03366	0.063	0.05829	0.06798	0	0	0	1	0
6	-0.03366	0.063	-0.05829	0.06798	0	0	0	1	0
7	0	0.063	0	0.06798	0.40151	0	0	1	0
8	0	0.063	0.11658	0.06798	0.40151	0	0	1	0

Figure 9: Example of a Particle Custom Inlet definition file shown in Microsoft Excel

Besides accepting only these file types, the other requirements for the definition file are that it is saved so that the values are separated by commas (sometimes called **comma delimited** or **Comma Separated Value** files) (Figure 8), and that it is formatted according to the way Rocky expects. Specifically, that the first row must contain the header information of the required and optional columns you are including, and that each individual particle making up the Particle Custom Inlet is defined in a separate subsequent row according to the following format, which is composed of fields that can be defined in any order:

`x,y,z,release,size,angle,nx,ny,nz,temperature,ux,uy,uz,wx,wy,wz`

These fields are explained in the following table.

Table 3.2: Particle Custom Inlet Definition File Field Definitions

Field	Type	Definition	Unit	Notes and Tips
xyz	Required	Defines the global coordinates of	mmm	• The particle will be positioned in this exact location based upon its geometric center.

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

Field	Type	Definition	Unit	Notes and Tips
		the domain location where the particle will be positioned.		• To help you prescribe the desired position, you can use the minimum and maximum coordinate values shown in the **Particles Details** window, keeping in mind that the shape's geometric center is in the (0,0,0) position relative to the local coordinate system of the particle.
release	Optional	The simulation time you want the particle to appear in the simulation.	s	• If you choose not to include this field, Rocky will assume a release time of 0 (zero) seconds. • As with the **Particle Inlet** method, if there are other particles or boundaries obstructing the particle at the release time and location defined, Rocky will delay its release and a warning will be displayed in the **Simulation Log**. • If you intend to use this Particle Custom Inlet with frozen Fibers , you must ensure that either all values for this field are set to 0 (zero), or this field is not included in the definition file.
size	Optional	Defines the **Size**, **Diameter**, or **Scale Factor** of the particle to be released, based upon the **Size Type** that you defined for the Particle set on the **Particle** | **Size** tab.	m	• This field applies only to single-element (rigid) particles. • **Important**: This field must not be included for multi-element (meshed) particles. Rocky will instead use for all released particles the single **Size** (**Diameter** or **Scale Factor**) value you set for the Particle set itself. • Similarly, if you choose not to include this field in your definition file for single-element (rigid) particles, Rocky will use for all released particles only the largest **Size** (**Diameter** or **Scale**

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

Field	Type	Definition	Unit	Notes and Tips
				Factor) value you set for the Particle set itself. • If you do choose to define this field for single-element (rigid) particles in your definition file, then the **Size** (**Diameter** or **Scale Factor**) value(s) you set for the Particle set itself (from its **Particle** | **Size** tab) will be ignored.
angle nx, ny, nz.	Optional	Defines the amount of particle rotation (angle) around a unit vector (nx,ny,nz).	rad mmm	• These values are the same as the **Base Orientation** parameters you might set when defining a Particle set on the **Particle** | **Orientation** tab. • As with the **Rotation Vector** values you set for the Particle set itself (from its **Particle** | **Orientation** tab), the X, Y, and Z components that define the vector will be normalized. • If you choose not to include this field, Rocky will assume no particle rotation. • If you choose to define these fields in your definition file, then the **Rotation Vector** and **Rotation Angle** values you set for the Particle set itself (from its **Particle** | **Orientation** tab) will be ignored.
temperature	Required only when the **Thermal Model** is enabled	Defines the temperature the particle will have at time of release.	K	• Values must be above but not equal to 0 K (zero Kelvin). • If this field is not defined in your custom input file but you have the **Thermal Model** enabled for your simulation, you will still be able to define a single initial temperature for all particles in the set by using the **Default Temperature** parameter. (See Table below.)

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

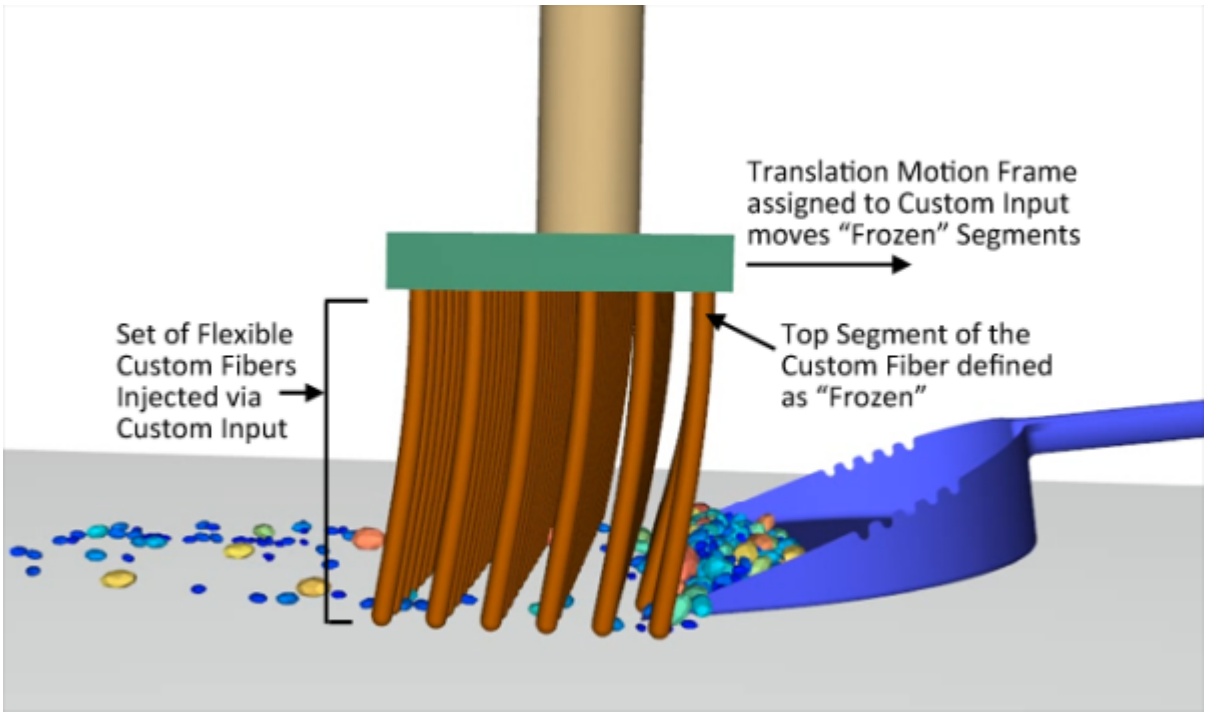
Field	Type	Definition	Unit	Notes and Tips
ux, uy, uz.	Optional	Defines the initial translational velocity of the particle as it is released.	m/sm/sm/s	
wx,wy,wz.	Optional	Defines the initial angular velocity of the particle as it is released.	rad/srad/srad/s	

After the definition file is imported as part of the **Particle Custom Inlet** process, Rocky will display a summary view of the file contents in a special **File Info** section (Figures 16-17). Here you can verify the number of particles that have been defined, as well as the minimum and maximum values of the other fields within the file.

PARTICLE CUSTOM INLETS AND FROZEN FIBERS

When combined with frozen Fibers (p. 1391) (see also About Defining and Importing Custom Particle Shapes (p. 670)), Particle Custom Inlets have the added benefit of assigning a Motion Frame (About Creating and Applying Motion Frames (p. 533)) to the injected particles. In these cases, the frozen Segments of the injected fibers will move according to the frame definition. This combination can be useful for simulating a broom, for example.

Figure 3.165: Broom example simulated with Particle Custom Inlets and Frozen Fibers



Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information
of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

In the example shown in Figure 10, the Particle Custom Inlet defines the exact placement of the flexible Custom Fiber bristles, and the assigned Motion Frame translates the fibers by moving only the frozen top Segment that was defined for the fibers.

Important:

For Particle Custom Inlets to work property with Frozen Fibers, all `release` times in the Particle Custom Inlet definition file must either be defined as 0 (zero) or the `release` field itself must be omitted from the file.

Limitations: In this release of Rocky, the three-way combination of a (1) Particle Custom Inlet being defined with (2) Frozen Fibers and also having a (3) Motion Frame assigned to the Particle Custom Inlet is, as a specific combination, incompatible with both Cartesian and Cylindrical Periodic Domains. (See also About Domain Settings Parameters (p. 452) and Particle and Inlet Limitations (p. 59).)

ABOUT THE INFO TAB

You can also use the information on the **Info** tab to get an estimate of how many particles might be released from each inlet during the simulation. (See also About the Info Tab (p. 83).)

Notes:

- For both Continuous Feed and Volumetric Inlet types, the particle estimate does not take into account simulation-specific events such as the inlet being blocked by a moving boundary or by previously-injected particles.
- For Volumetric Inlet specifically, the estimate assumes there are no boundary limits to consider, and assumes a **Gap Scale Factor** equal to **1**. This means that if you have set your particle Mass higher than the box and gap you define can contain, the particle estimate on the Info tab will show a higher number than what will actually be injected (Figure 3).
- For Particle Custom Inlet specifically, the particle estimate is equal to the **Number of Particles** value on the **Particle Custom Inlet** tab.

Processing Requirements

Before you can process your simulation, you must create at least one particle or fluid inlet and provide either the Mass Flow Rate value (for Fluid or Particle Inlet), Mass value (for Volumetric Inlet), or valid definition file (for Particle Custom Inlet) for at least one Particle set.

Inlets and Outlets and External Modules

If you have enabled one or more external Modules (p. 1391) that affect your Inlet or Outlet settings, you might also have a separate **Modules** sub-tab or separate, module-specific settings that you can define. Refer to the Module's documentation (if provided) for more information. (See also Rocky Simulation Entities that can be Affected by Modules (p. 223).)

Inlets and Outlets Parameter Definitions

Use the figures and tables below to help you understand the various parameters you can set for inlets.

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

Figure 3.166: Particle Inlet, Particles sub-tab, Data Editors panel

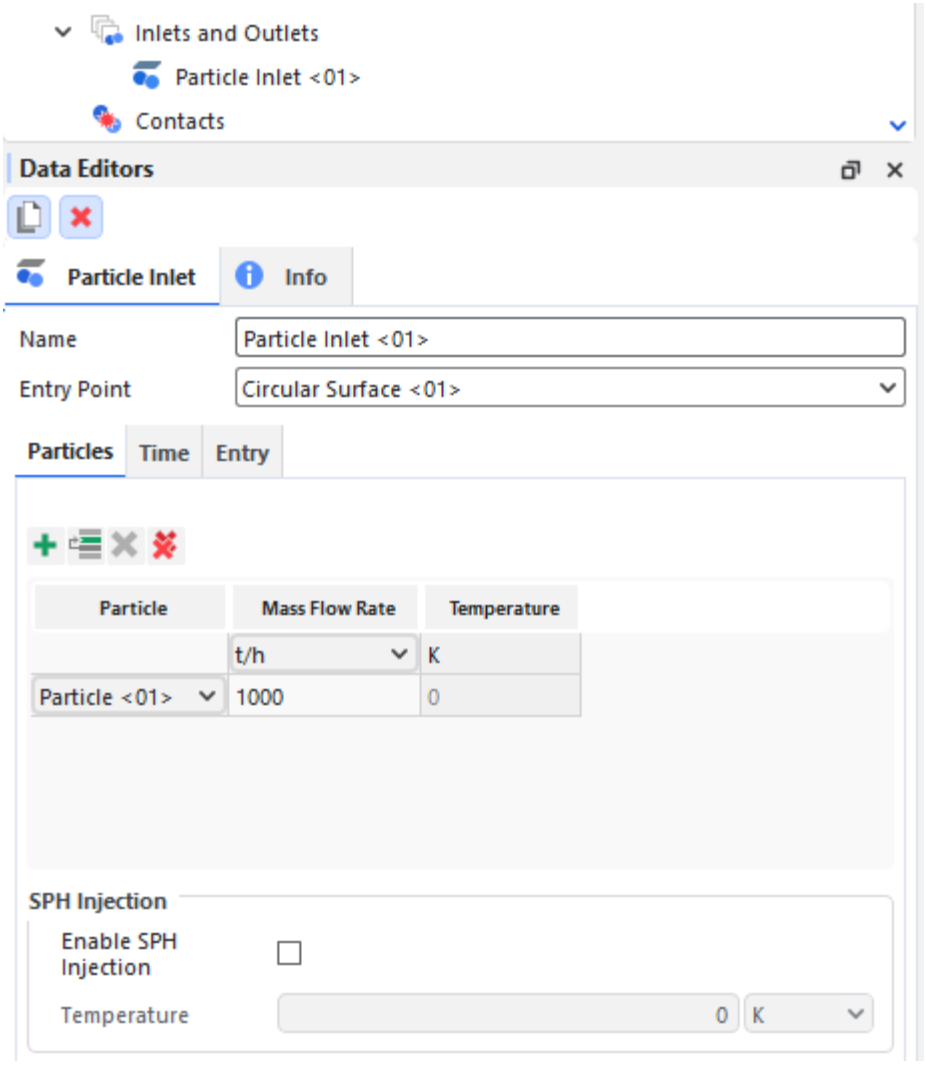
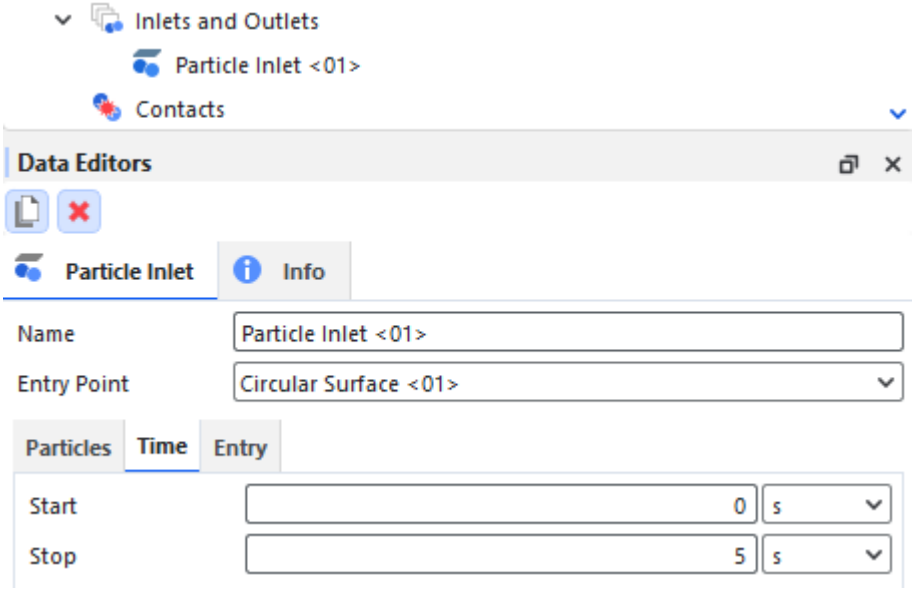


Figure 3.167: Particle Inlet, Time sub-tab, Data Editors panel



Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

Figure 3.168: Particle Inlet, Entry sub-tab, Data Editors panel

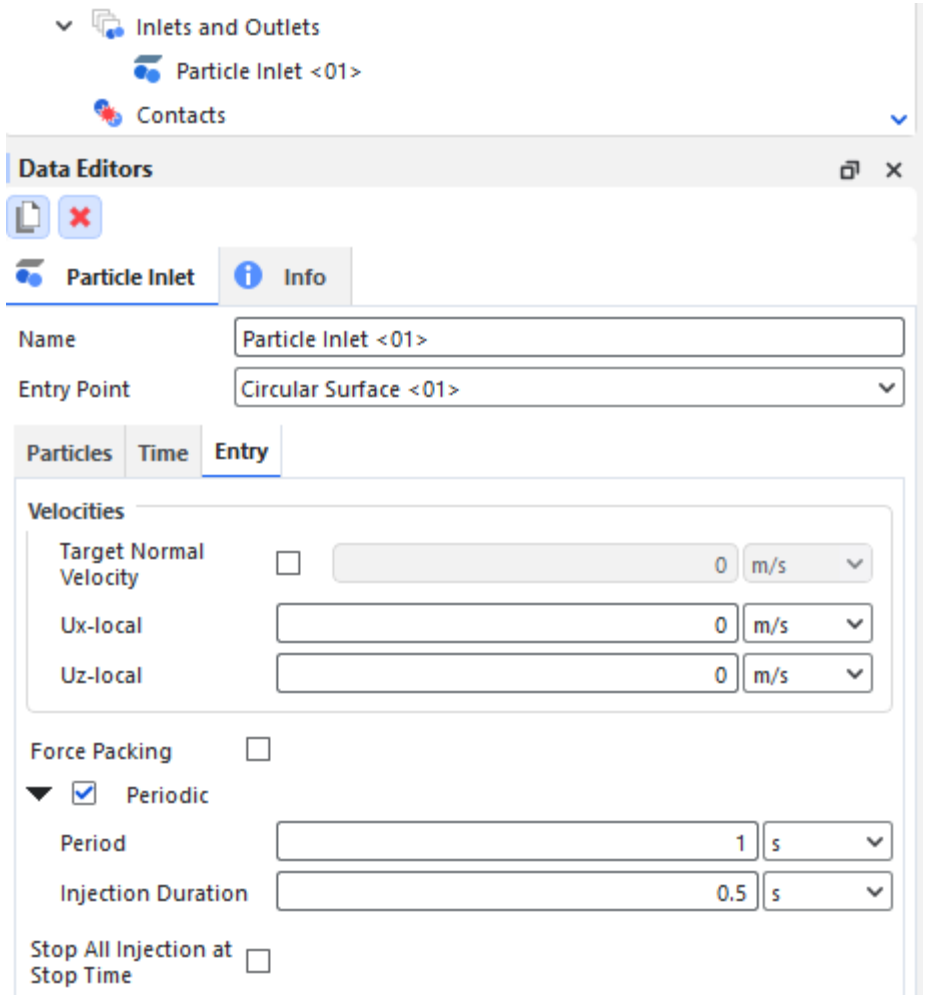


Table 2: Particle Inlet parameter options

Setting	Description	Range
Name	Enables you to specify a unique identifier for the particle inlet.	99 character limit
Entry Point	The Feed Conveyor or inlet from which the particles are released during the simulation. **Note:** Only one Inlet can be set per Entry Point. **Tip:** To have multiple Inlets be released from a single location, create multiple Inlets and then define a separate Input per Inlet.	Any Planar Surface

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

Setting	Description	Range
Particles sub-tab		
Particle	The Particle set (p. 1393) that you want to enter the simulation from the **Entry Point** specified.	Any Particle set specified in the simulation
Mass Flow Rate	The mass flow rate for the **Particle** set that you want released from this **Entry Point**.	Values above but not equal to 0 ton/h **Note:** The one exception to this rule is if you want to change the active Inlets after you Save a Copy of a Partially Processed Simulation for Restart Purposes (p. 855). (See also About Changing Geometries and/or Particles in a Partially Processed Simulation (p. 847)).
Temperature	When **Thermal Model** is enabled (see also About Physics Parameters (p. 417)), this is the temperature that all particles in the **Particle** set will have when released from this **Entry point** .	Values above but not equal to 0 K (zero Kelvin)
Enable SPH Injection	When the check box is enabled, allows for the injection of fluid that fills the space between the particles.	Turns on or off
Temperature	The temperature of the fluid that will be inject with the particles (Only available if Enable SPH Injection checkbox and the **Thermal Model** are enabled).	Values above but not equal to 0 K (zero Kelvin)
Time sub-tab		

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

Setting	Description	Range
Start	The time you want particles to begin releasing from the **Entry Point**. Note: If you choose to use parametric expressions in this field, know that only the resulting value and not the variables and/or mathematical functions you enter will be retained in any project copies you save for restart purposes. (See also I get a "Links removed" message when I save my project for restart purposes (p. 1381)).	Positive values **Tip:** Check the Status panel to ensure that any variables or mathematical functions you might use results in valid values. (See also Double-Click the Status Panel to Jump to the Appropriate UI Location (p. 117)).
Stop	The time you want particles to stop releasing from the **Entry Point**. Note: If you choose to use parametric expressions in this field, know that only the resulting value and not the variables and/or mathematical functions you enter will be retained in any project copies you save for restart purposes. (See also I get a "Links removed" message when I save my project for restart purposes (p. 1381)).	Positive values **Tip:** Check the Status panel to ensure that any variables or mathematical functions you might use results in valid values. (See also Double-Click the Status Panel to

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

Setting	Description	Range
		Jump to the Appropriate UI Location (p. 117)).
Entry sub-tab		
Target Normal Velocity	When selected, enables you to set the normal speed of the particles being released from the **Entry Point**. The speed is measured in local coordinates.	Positive values (but limited by flow rate)
Ux-local	Horizontal speed of the particles being released from the **Entry Point**. Measured in local coordinates.	No limit
Uz-local	Out-of-plane speed of the particles being released from the **Entry Point**. Measured in local coordinates.	No limit
Force Packing	When the **Advanced Features** checkbox is enabled on the **Options | Preferences** dialog (see also About Setting Global Preferences (p. 227)), this determines how packed together particles will be when released from the **Entry Point**. When enabled, Rocky ignores the particle boundary overlaps when determining when to release particles, which for especially long or randomly oriented particles, can result in particles being released in closer proximity to each other. When cleared, Rocky will respect the particle boundary overlaps when determining when to release particles, which for especially long or randomly oriented particles, can result in extra wide gaps between particles when they are released.	Turns on or off
Periodic	When selected, enables particles to be released from the **Entry Point** in periodic bursts. When cleared, particles will be released in a continuous flow. Rocky calculates the particle mass released in each periodic burst by equally dividing the total particle mass by the total number of bursts during the simulation. The total particle mass is defined by the following two items: • The total time the inlet is in active use, which takes into account the **Simulation Duration** (see also About Solver Parameters (p. 464)) and the inlet's **Start** and **Stop** values.	Turns on or off

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

Setting	Description	Range
	• The **Mass Flow Rate** values for the inlet. Rocky calculates the number of bursts that will occur throughout the simulation based upon the following three items: • The total time the inlet is in active use, which takes into account the **Simulation Duration** (see also About Solver Parameters (p. 464)) and the inlet's **Start** and **Stop** values. • The total injection **Period**. • The active **Injection Duration** of the period. *For example*, if you set up a particle inlet with a **Period** of 3 s, an **Injection Duration** of 1 s, and have a total simulation time of 3.5 s, the particles will be released in two bursts: one (complete) burst starting at t=0 s and stopping at t=1 s, and a second (incomplete) burst starting at t=3 s and stopping at t=3.5 s.	
Period	When **Periodic** is enabled, this defines the length of time for each injection cycle.	Positive values
Injection Duration	When **Periodic** is enabled, this defines the amount of time during each **Period** when particles will be actively released.	Positive values Note: Must be less than or equal to the **Period** value.
Stop All Injection at Stop Time	In cases where Rocky delays particle release due to other particles or boundaries being in the way, selecting this item ensures that particles stop releasing at the **Stop** time. If this item is cleared, particles will stop releasing when the originally calculated number of particles have been released.	Turns on or off
Modules sub-tab		
Particle	For each row, lists the Particle set (p. 1393) that is defined on the **Particles** sub-tab.	Automatically provided
(Varies)	These settings are specific to only certain external Module (p. 1391) and are therefore not documented in the Rocky User Manual. Refer to the Module's documentation (if provided) for more information. (See also Rocky Simulation Entities that can be Affected by Modules (p. 223)).	(Varies)

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

Figure 3.169: Fluid Inlet, SPH sub-tab, Data Editors panel

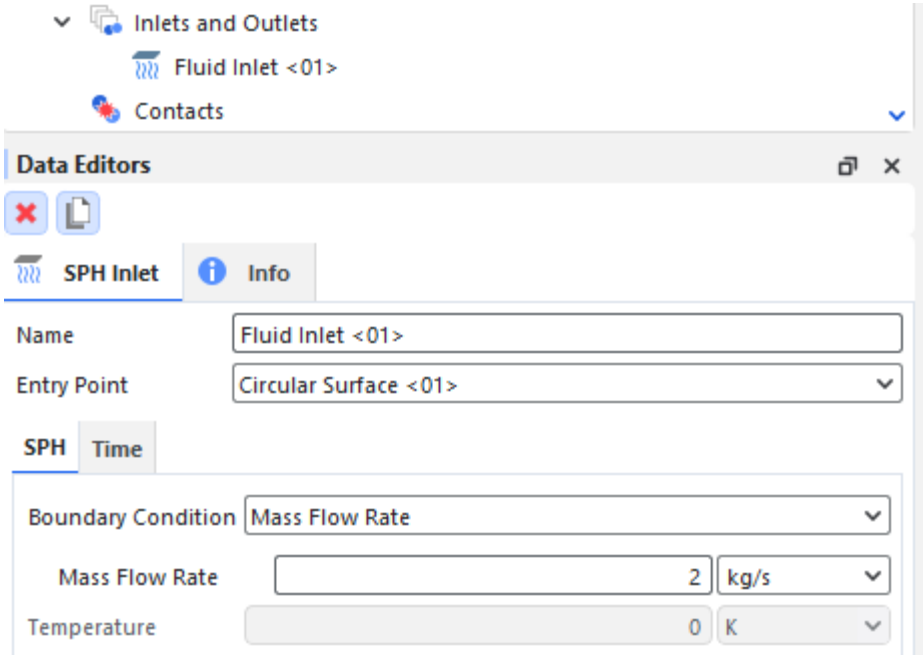


Figure 3.170: Fluid Inlet, Time sub-tab, Data Editors panel

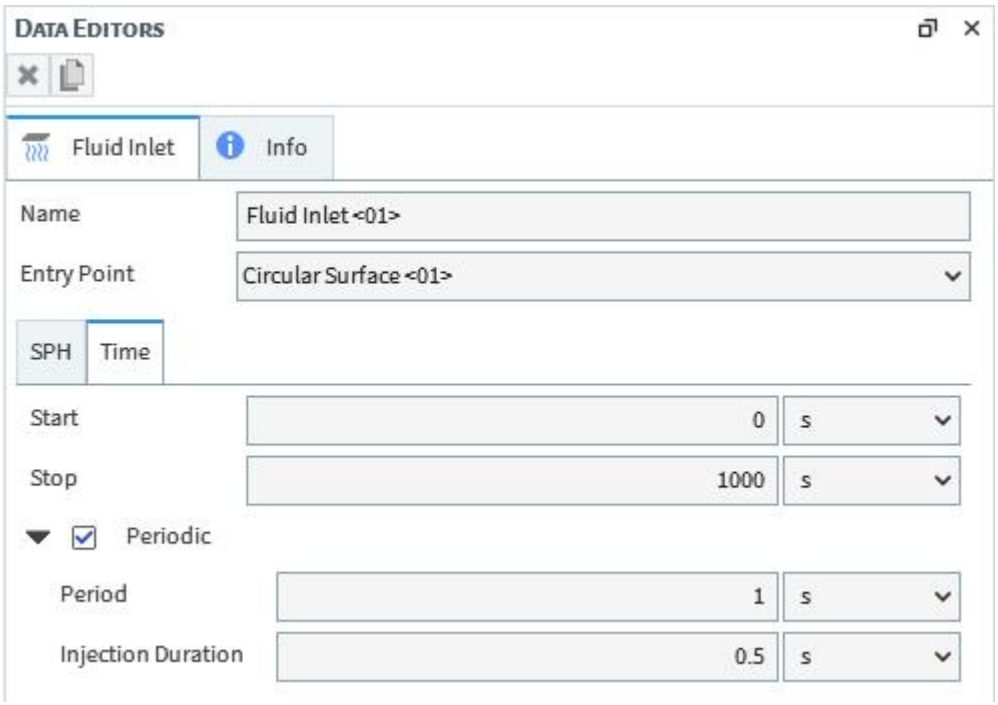


Table 3.3: Fluid Inlet parameter options

Setting	Description	Range
Name	Enables you to specify a unique identifier for the fluid inlet.	99 character limit

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

Setting	Description	Range
Entry Point	The surface inlet from which the fluid is released during the simulation. **Note:** Only one Inlet can be set per Entry Point.	Any Planar Surface
SPH		
Mass Flow Rate	The mass flow rate forthe Fluid elements that you want released from this **Entry Point**	Values above but not equal to 0 kg/s **Note:** The one exception to this rule is if you want to change the active Inlets after you save a copy of a partially processed pimulation for pestart purposes.
Temperature	When Thermal Model is enabled this is the temperature that all particles in the **Particle**set will have when released from this **Entry point.**	Values above but not equal to 0 K (zero Kelvin)
Time sub-tab		
Start	The time you want fluid elements to begin releasing from the **Entry Point**.	Positive values **Note:** Check the Status panel to ensure that any variables or mathematical functions you might use results in valid values.

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

Setting	Description	Range
Stop	The time you want fluid elements to stop releasing from the **Entry Point.** **Note:** If you choose to use parametric expressions in this field, know that only the resulting value and not the variables and/or mathematical functions you enter will be retained in any project copies you save for restart purposes.	Positive values **Note:** Check the Status panel to ensure that any variables or mathematical functions you might use results in valid values.
Periodic	Enables the Time Periodic control for SPH Fluid Inlets, creating a series of inlets to mimic periodic behavior.	Enable or Disable
Period	Defines the Time of the all period will last.	Positive values
Injection Duration	Defines the fluid injection duration in the already defined.	Positive values

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

Figure 3.171: Volumetric Inlet, Particles sub-tab, Data Editors panel

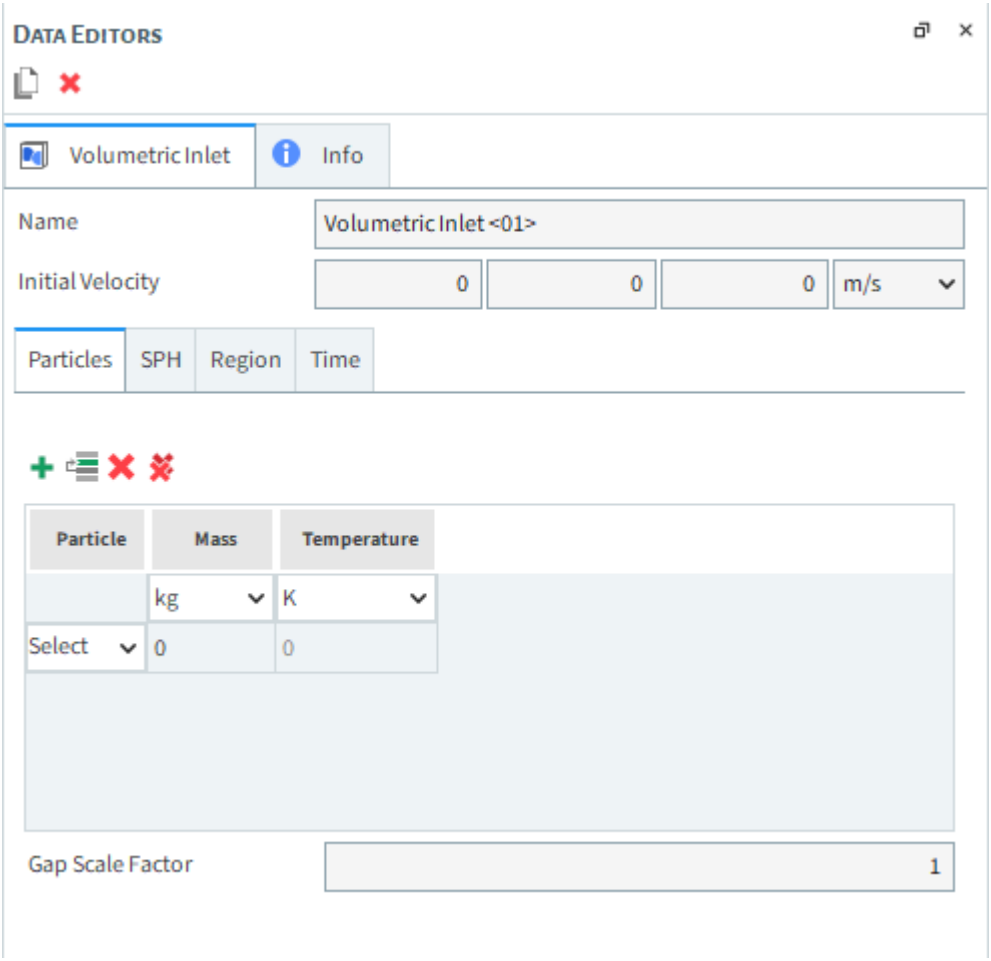
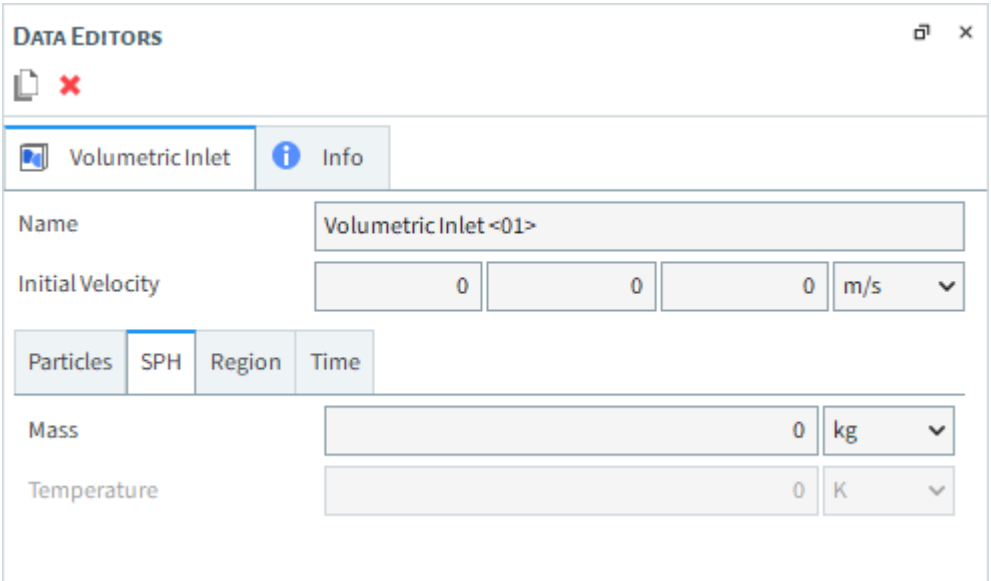
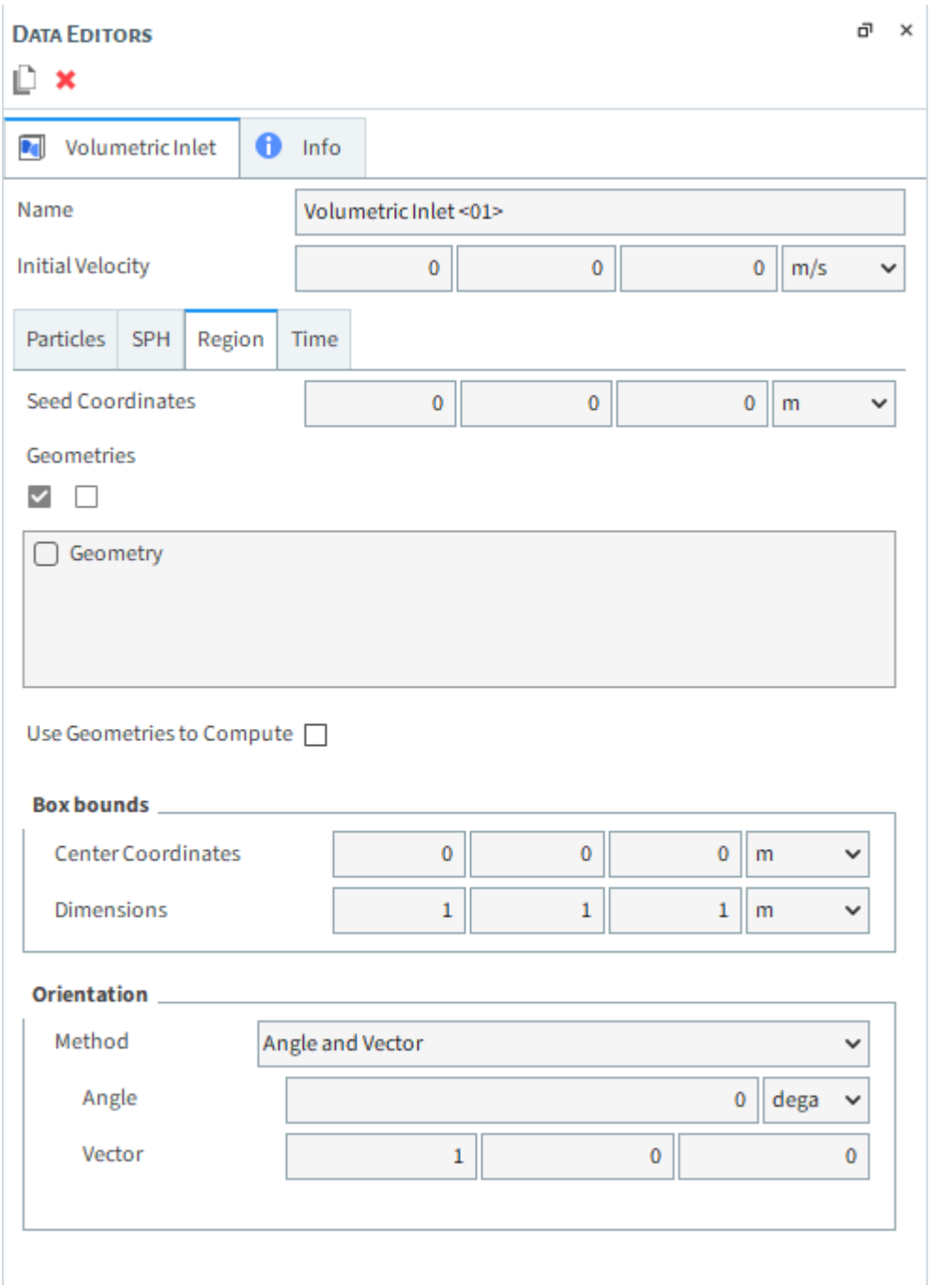


Figure 3.172: Volumetric Inlet, SPH sub-tab, Data Editors panel



Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

Figure 3.173: Volumetric Inlet, Region sub-tab, Data Editors panel



Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

Figure 3.174: Volumetric Inlet, Time sub-tab, Data Editors panel

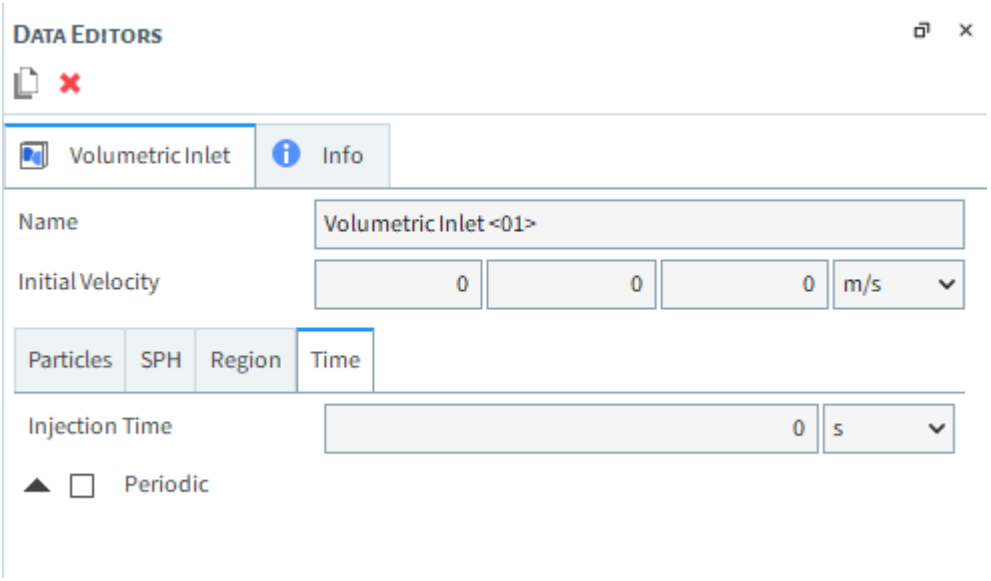


Table 4: Volumetric Inlet parameter options

Setting	Description	Range
Name	Enables you to specify a unique identifier for the inlet.	99 character limit
Initial Velocity	Enables you to set a vector of initial velocities in the x, y and z-axis for both DEM particles and SPH elements. **Important:** The initial velocities vector is the same for all DEM particles and all SPH elements defined on the same volumetric inlet. **Note:** The initial velocities vector cannot be changed for resume or restart simulations.	Any values
Particles sub-tab		
Particle	The Particle set (p. 1393) that you want to enter into simulation by building around the **Seed Coordinate**, which is itself specified on the **Region** sub-tab.	Any Particle set specified in the simulation

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

Setting	Description	Range
Mass	The target mass of particles within the set that you want built around the **Seed Coordinate**, which is itself specified on the **Region** sub-tab. **Tip:** To avoid entering a value that is too large to fully simulate, ensure that you are entering a Mass value appropriate for the Particle set and Box volume you specify on the **Region** sub-tab.	Values above but not equal to 0 kg
Temperature	When **Thermal Model** is enabled (see also About Physics Parameters (p. 417)), this is the temperature that all particles in the Particle set will have when injected from this inlet method.	Values above but not equal to 0 K (zero Kelvin)
SPH sub-tab		
Mass	The target mass of fluid within the set that you want built around the **Seed Coordinate**, which is itself specified on the **Region** sub-tab. **Tip:** To avoid entering a value that is too large to fully simulate, ensure that you are entering a Mass value appropriate for the box volume you specify on the **Region** sub-tab.	Values above but not equal to 0 kg
Temperature	When **Thermal Model** is enabled (see also About Physics Parameters (p. 417)), this is the temperature that the fluid elements will have when injected from this inlet method.	Values above but not equal to 0 K (zero Kelvin)
Region sub-tab		
Seed Coordinates	The X, Y, and Z coordinate location of the point ("Seed") around which the particle layers will be built. **Tip:** To avoid particle generation issues, ensure that you locate your **Seed Coordinate** away from the boundaries of your simulation geometries.	No limit but the location must be within the limits of the Volumetric Inlet **Box bounds**

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

Setting	Description	Range
Gap Scale Factor	Determines how closely particles are positioned to each other (i.e., how much of a "gap" there is between them) during injection. Specifically: • A value of **1** will inject the particles right next to each other. (No gap.) • A value **greater than 1** will increase the gap between particles during injection. Gap size is based upon how the particles would have been injected if their size was increased by the factor entered. For example, entering a **Gap Scale Factor** of **2** would inject the particles as if they were twice the size, which provides a larger gap between each particle than the default setting.	1 or greater
Geometries	Lists the imported geometry components within the simulation by which you are able to constrain your Volumetric Inlet. Specifically: • If no geometries are selected, the Volumetric Inlet will use only the **Box bounds** information you enter to constrain the injection of particles. This means that if there are geometries within the **Box bounds**, they will be ignored until after the particles are released. • If one or more geometries are selected, the Volumetric Inlet will consider those boundaries-as well as the **Box bounds**-during the actual injection of particles. This means that if there are geometries within the **Box bounds**, they will be respected when the Volumetric Inlet is built. In addition, selecting a geometry will also enable you to **Use Geometries to Compute** the **Box bounds**.	Turns off or on
Use Geometries to Compute	When one or more geometry components are selected from the **Geometries** list, selecting this option enables Rocky to automatically define the **Box bounds** based upon the limits of the selected geometries. When this option is cleared, Rocky will use the **Box bounds** you define.	Turns off or on
Box bounds Center Coordinates	When the **Use Geometries to Compute** checkbox is cleared, this defines the X, Y, and Z coordinate location of the boundary box's center point. (The bounds of this box is what will constrain your Volumetric Inlet.)	No limit
Box bounds Dimensions	When the **Use Geometries to Compute** checkbox is cleared, this defines the X, Y, and Z dimensions of the box that will constrain your Volumetric Inlet.	No limit

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

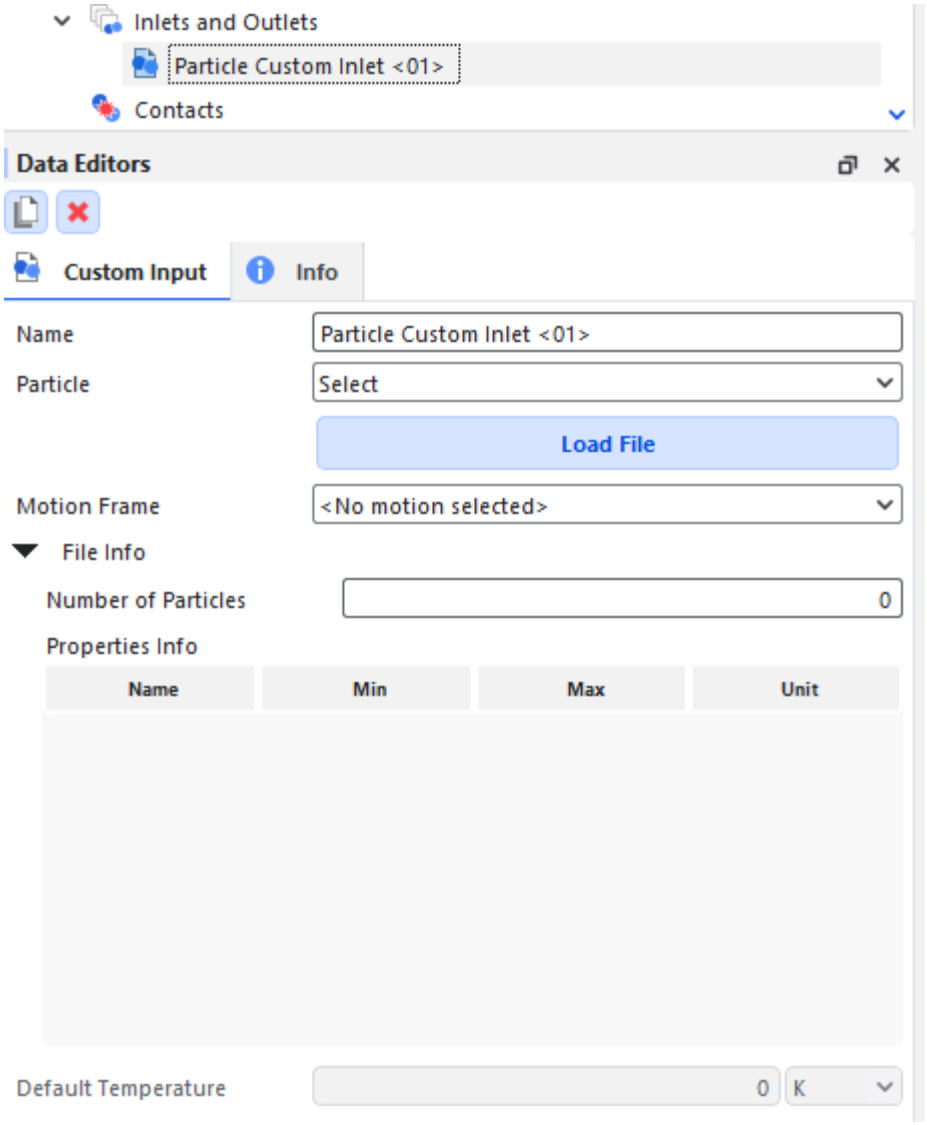
Setting	Description	Range
Orientation Method	Enables you to select how you want to define the orientation of the Volumetric Inlet. Specifically: • **Angle and Vector** enables you define one vector and one angle of rotation around it. • **Basis Vectors** enables you to define the X, Y, and Z directions of the Volumetric Inlet local basis. • **Angles** enables you to define angles of rotation in three directions, the order of which you can also specify.	Angle and Vector; Basis Vectors; Angles
Orientation Method **Angle and Vector**		
Angle	When **Angle and Vector** is selected for **Orientation**, this is the angle the Volumetric Inlet will rotate around the **Vector** defined.	Any value
Vector	When **Angle and Vector** is selected for **Orientation**, this is the X, Y, and Z components that define the vector around which the Volumetric Inlet will rotate, using the **Angle** defined.	No limit but the values entered will be normalized
Orientation Method **Basis Vectors**		
X-Direction	When **Basis Vectors** is selected for **Orientation**, this is the coordinate value defining the first of three directional vectors that together define the final orientation of the Volumetric Inlet. **Tip:** To ensure correct results, make sure you define this vector as orthogonal (perpendicular) to the other two vectors.	No limit but the values entered will be normalized
Y-Direction	When **Basis Vectors** is selected for **Orientation**, this is the coordinate value defining the second of three directional vectors that together define the final orientation of the Volumetric Inlet. **Tip:** To ensure correct results, make sure you define this vector as orthogonal (perpendicular) to the other two vectors.	No limit but the values entered will be normalized

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

Setting	Description	Range
Z-Direction	When **Basis Vectors** is selected for **Orientation**, this is the coordinate value defining the third of three directional vectors that together define the final orientation of the Volumetric Inlet. **Tip:** To ensure correct results, make sure you define this vector as orthogonal (perpendicular) to the other two vectors.	No limit but the values entered will be normalized
Orientation Method **Angles**		
Order	When **Angles** is selected for **Orientation**, this defines the order in which the three **Rotation** fields will be applied.	XYZ; ZXY; YXZ; YZX; XZY; ZYX
Local Angles	When **Angles** is selected for **Orientation**, this defines what coordinate system will be used as a basis for the angle specified. Specifically: • When enabled, the angle will be based on the local coordinate system. • When cleared, the angle will be based on the global coordinate system.	Turns on or off
Rotation	When **Angles** is selected for **Orientation**, this is the degree of Volumetric Inlet rotation in each of the three directions specified by the **Order** provided.	Any value
Time sub-tab		
Injection Time	The simulation time at which you set the volumetric injection to occur.	Positive values
Periodic	Enables periodic volumetric injection of Particles and/or Fluid into the simulation.	Turns on or off
Period Time	The period at which you set the volumetric injection to occur.	Positive values
Stop Time	The simulation time at which you set the periodic injection to stop.	Positive values
Modules sub-tab		
Particle	For each row, lists the Particle set (p. 1393) that is defined on the **Particles** sub-tab.	Automatically provided
(Varies)	These settings are specific to only certain external Module (p. 1391) and are therefore not documented in the Rocky User Manual. Refer to the Module's documentation (if provided) for more information. (See also Rocky Simulation Entities that can be Affected by Modules (p. 223)).	(Varies)

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

Figure 3.175: Particle Custom Inlet, Data Editors panel



Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

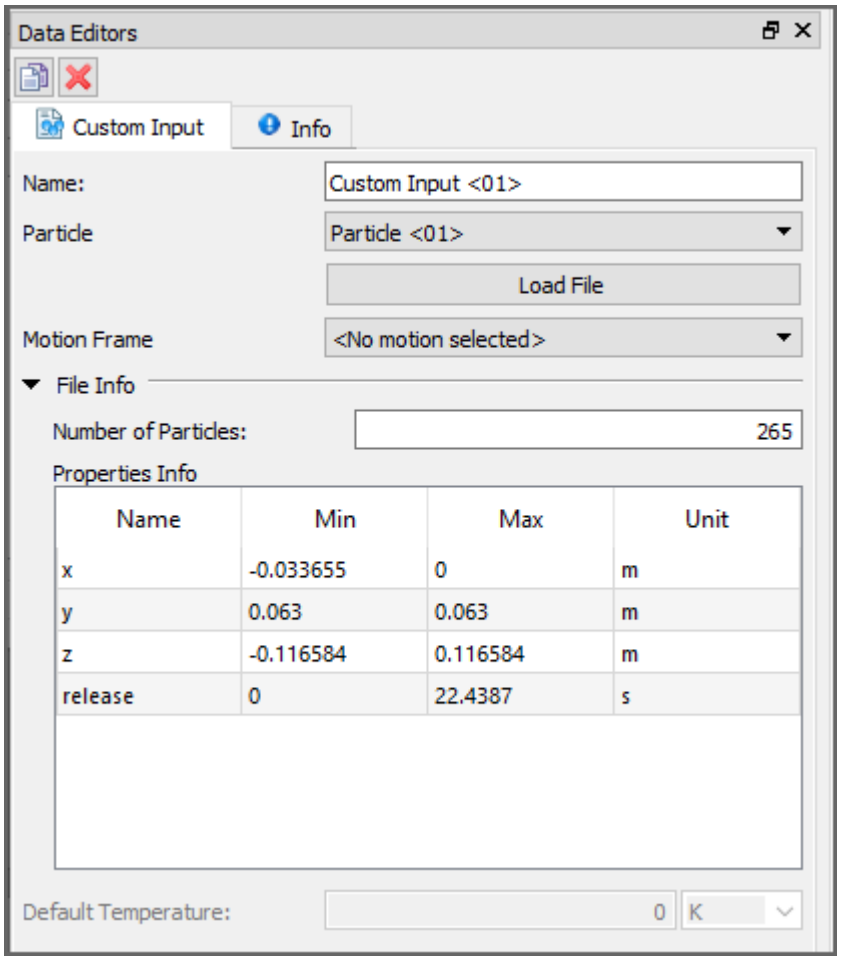


Figure 20: Particle Custom Inlet, Data Editors panel when a multi-element (flexible and/or discrete-breakable) Fiber with frozen elements is defined

---

Note:

The Overlap Check for Particle Custom Inlet injection only works when the particle is released at t>0s. If the particle is released at t=0s, this overlap check is not performed.

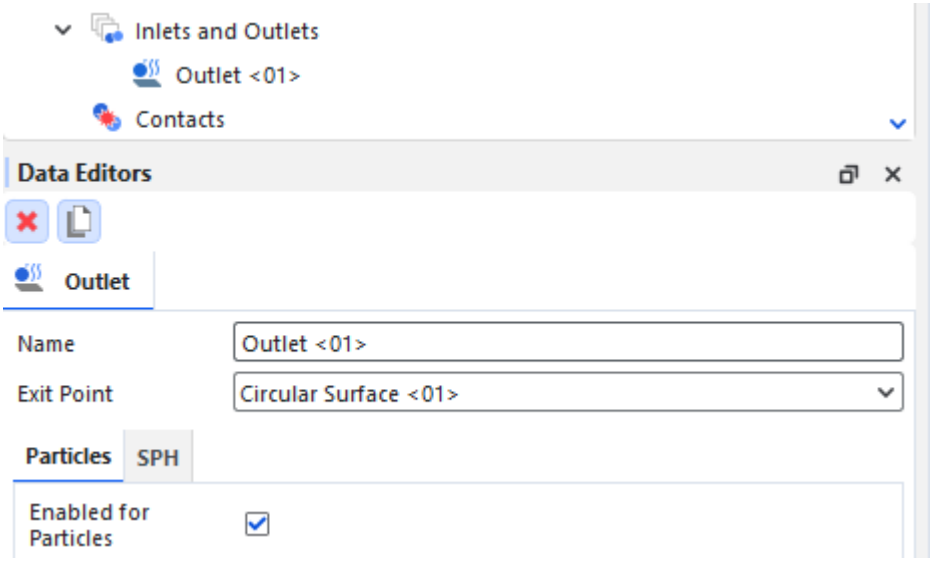
Table 5: Particle Custom Inlet parameter options

Setting	Description	Range
Name	Enables you to specify a unique identifier for the particle inlet.	99 character limit
Particle	Enables you to select which of the available particle sets to apply to the Particle Custom Inlet.	Automatically determined
Motion Frame	When a multi-element (flexible and/or breakable) Custom Fiber with frozen elements defined is selected for the **Particle**, this defines which Motion Frame you want assigned to the frozen elements of the injected fibers.	Automatically determined
Number of Particles	Lists the total amount of particles that will be released based upon this inlet into the simulation. This number is	Display only

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

Setting	Description	Range
	based upon the rows defined in the imported Particle Custom Inlet file.	
Properties Info	Displays a table of summarized details based upon the Particle Custom Inlet file that was imported. This data includes the field **Name**, which corresponds to the column header used within the file; the **Min** and **Max** values, which together summarize the range of values being used within the particles defined; and the **Unit**, which displays the hard-coded unit of measurement always used when defining that particular field type.	Display only
Default Temperature	When the **Thermal Model** is enabled (see also About Physics Parameters (p. 417)) and no **temperature** values were defined in the imported Particle Custom Inlet file, this is the temperature that all particles in the **Particle** set will have at the time of release.	Values above but not equal to 0 K (zero Kelvin)
Other "Modules" Settings		
(Varies)	These settings are specific to only certain external Module (p. 1391) and are therefore not documented in the Rocky User Manual. Refer to the Module's documentation (if provided) for more information. (See also Rocky Simulation Entities that can be Affected by Modules (p. 223)).	(Varies)

Figure 3.176: Outlet, Particles tab



Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

Figure 3.177: Outlets, SPH tab

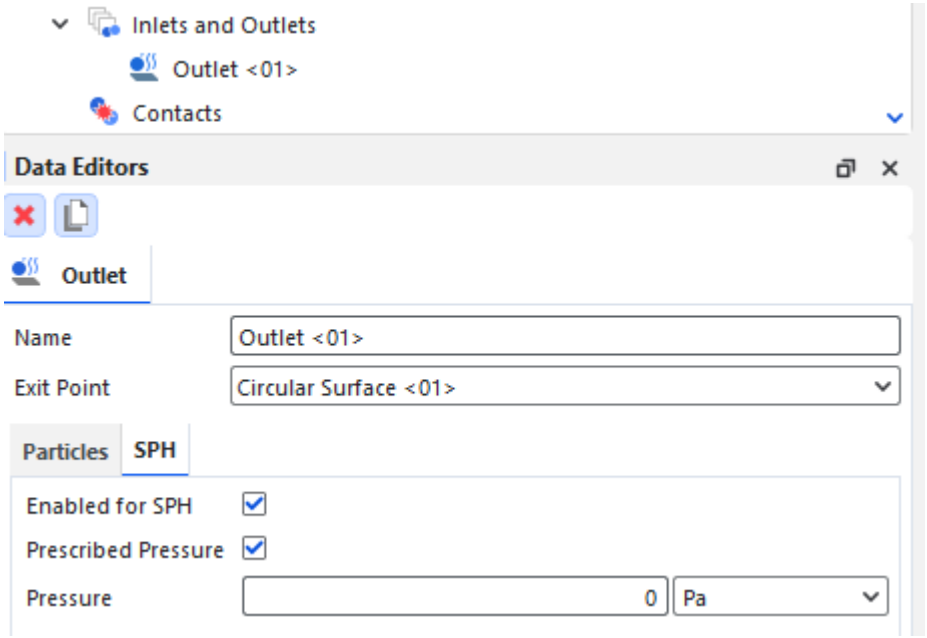


Table 6: Outlet parameter options

Setting	Description	Range
Name	Enables you to specify a unique identifier for the outlet.	99 character limit
Exit Point	The surface that will act as an exit point for both, fluid and particles, during the simulation.	Any Planar Surface
Particles sub-tab		
Enabled for Particles	Specifies if the outlet will act as an **Exit Point** for **Particles**.	Turns off or on
SPH sub-tab		
Enabled for SPH	Specifies if the outlet will act as an **Exit Point** for **Fluid**.	Turns off or on
Prescribed Pressure	Specifies if the outlet will use a Prescribed Pressure as a boundary condition.	Turns off or on
Pressure	Defines the pressure that will be applied to the Prescribed Pressure boundary condition.	Any Value

What would you like to do?

- Create a New Particle Inlet (p. 740)
- Create a New Fluid Inlet (p. 741)
- Create a New Volumetric Inlet (p. 741)
- Create a New Particle Custom Inlet (p. 743)

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

- Edit an Inlet or an Outlet (p. 745)
- Remove an Inlet or an Outlet (p. 745)
- Learn more About the Info Tab (p. 83)

CREATE A NEW PARTICLE INLET

1. From the **Data** panel, click **Inlets and Outlets**, and the from the **Data Editors** panel, click the **Create Particle Inlet** button. A new Particle Inlet component appears under **Inlets and Outlets** in the **Data** panel. Tip:You may also access this functionality from the right-click menu on the **Data** panel.
2. From the **Data** panel, click the Particle Inlet component you just added and then from the **Data Editors** panel, on the main **Particle Inlet** tab, enter the **Name** and set the **Entry Point** you want.
3. From the **Particles** sub-tab, do all of the following:
   a. Click the **Add** button. A new row appears at the bottom of the table.
   b. Under **Particle**, choose the Particle set that you want, and then specify the **Mass Flow Rate** and **Temperature** (if applicable) you want associated with that Particle set.
   c. Repeat step 3 for each Particle set that you want entering the simulation from the **Entry Point** specified.
4. If desired, select the check box to allow for fluid elements inlet alongside the DEM particles and define the **Temperature** (if aplicable).
5. From the **Time** sub-tab, define what you want for **Start** and **Stop** times.
6. If desired, from the **Entry** sub-tab, define the **Velocities** parameters you want. Optionally, you can also define the **Periodic** injection settings you want.
7. If you have enabled a Module that has unique Input properties (see also About Modules Parameters (p. 431)), select the **Modules** sub-tab, and then for each **Particle** row specified, enter the value you want for the module-specific parameters provided.

Tips:

- To insert a new row beneath the selected row, click the **Insert** button.
- To remove the selected row, click the **Remove** button.
- To remove all rows in the table, click the **Remove All** button.
- To see an estimate of the number of particles that might be released from a Particle Inlet during the simulation, view the **Info** tab.

See Also:

- About Adding and Editing Particle Inlets (p. 708)
- Edit an Inlet or an Outlet (p. 745)

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

- Remove an Inlet or an Outlet (p. 745)
- About the Info Tab (p. 83)

CREATE A NEW FLUID INLET

1. From the **Data** panel, click **Inlets and Outlets**, and the from the **Data Editors** panel, click the **Create Fluid Inlet** button. A new Fluid Inlet component appears under **Inlets and Outlets** in the **Data** panel. Tip:You may also access this functionality from the right-click menu on the **Data** panel.
2. From the **Data** panel, click the Fluid Inlet component you just added and then from the **Data Editors** panel, on the main **Fluid Inlet** tab, enter the **Name** and set the **Entry Point** you want.
3. From the **SPH** sub-tab, do all of the following:
   a. Set the **Mass Flow Rate** at the inlet.
   b. If applicable, specify the fluid **Temperature** at the inlet.
4. From the **Time** sub-tab, define what you want for **Start** and **Stop** times.

See Also:

- Create a New Fluid Inlet (p. 741)
- Edit an Inlet or an Outlet (p. 745)
- Remove an Inlet or an Outlet (p. 745)

CREATE A NEW VOLUMETRIC INLET

1. From the **Data** panel, click **Inlets and Outlets**, and the from the **Data Editors** panel, click the **Create Volumetric Inlet** button. A new Volumetric Inlet component appears under **Inlets and Outlets** in the **Data** panel. Tip:You may also access this functionality from the right-click menu on the **Data** panel.
2. From the **Data** panel, click the Volumetric Inlet component you just added and then from the **Data Editors** panel, on the main **Volumetric Inlet** tab, enter the **Name** you want.
3. To add **Particles**: From the **Particles** sub-tab, do the following:
   a. Click the **Add** button. A new row appears at the bottom of the table.
   b. Under **Particle**, choose the Particle set that you want, and then specify the **Mass** and **Temperature** (if applicable) you want associated with that Particle set.
   c. Repeat steps 3a-3b for each Particle set that you want entering the simulation from the fill **Region** you will define.
4. To add **Fluid**: From the **SPH** sub-tab, do the following:
   a. Set the total fluid **Mass**.|br|
   b. If applicable, specify the fluid **Temperature**.

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

5. From the **Region** sub-tab, do the following:

   a. Next to **Seed Coordinates**, enter the X, Y, and Z coordinate location of the spherical point around which your Volumetric Inlet layers will be built. Important: This must be located within your **Box bounds**. Tip:Avoid locating this directly on the boundaries of your simulation geometries.

   b. If you want there to be some space between your particles when they are injected, enter a value above **1** for **Gap Scale Factor**. Otherwise, leave this setting as **1** to have particles injected closely together.

   c. Do one of the following:

      • To have Rocky automatically calculate your Box bounds based upon one or more imported geometries in your simulation, select the geometries you want from the **Geometries** list, and then select the **Use Geometries to Compute** checkbox.

      • To define your own box bounds but still have your Volumetric Inlet constrained to one or more geometry components, select the geometries you want from the **Geometries** list, and then define the **Center Coordinates** and **Dimensions** of your Box bounds.

      • To define your own Box bounds and have your Volumetric Inlet ignore any geometries within the bounds you define, keep both the **Geometries** and the **Use Geometries to Compute** checkboxes cleared, and then define the **Center Coordinates** and **Dimensions** of your Box bounds.

      • To define your Volumetric Inlet **Orientation**, keep the **Use Geometries to Compute** checkbox cleared, then select the **Method** and specify its parameters.

6. If you have enabled a Module that has unique Input properties (see also About Modules Parameters (p. 431)), select the **Modules** sub-tab, and then for each **Particle** row specified, enter the value you want for the module-specific parameters provided.

Tips:

• For the **Region** tab:

  – To visualize the Box bounds and Seed location of your Volumetric Inlet input, select or create a 3D View window and then from the **Data** panel, select the Volumetric Inlet entity you want to see. (See also View Geometries, Particles, Points, and Fluids in 3D (p. 1041).)

  – In the 3D View window, you can also move and resize the fill area by clicking and dragging the handles (colored dots) representing the center, and the local X, Y, and Z locations respectively. (See also Use Colored Directional Handles (p. 126).) Note: You still must move the Seed location by using the **Seed Coordinates**.

• For the **Particles** tab:

  – To insert a new row beneath the selected row, click the **Insert** button.

  – To remove the selected row, click the **Remove** button.

  – To remove all rows in the table, click the **Remove All** button.

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

- To see an estimate of the number of particles that might be released from a Particle Input during the simulation, view the **Info** tab. Notes:
  - This estimate assumes there are no boundary limits to consider and that the **Gap Scale Factor** is equal to **1**.
  - Ensure that the estimate includes enough particles; Volumetric Inlets require a minimum of 3 particles in order to create the "Seed". (See also Particle and Input Limitations (p. 59).)

See Also:

- About Adding and Editing Inlets and Outlets (p. 708)
- Create a New Particle Inlet (p. 740)
- Create a New Fluid Inlet (p. 741)
- Create a New Particle Custom Inlet (p. 743)
- Edit an Inlet or an Outlet (p. 745)
- Remove an Inlet or an Outlet (p. 745)
- About the Info Tab (p. 83)
- Use Colored Directional Handles (p. 126)

CREATE A NEW PARTICLE CUSTOM INLET

1. In a spreadsheet program or text editor, use the formatting requirements (see also About Adding and Editing Particle Inlets and Outlets (p. 708)) to define the fields and values you want for each particle you want injected, and then save the file as one of the supported CSV or Excel/ODF file formats (see also File Types and Folders in Rocky (p. 129)) ensuring that the file is saved so that the values are separated by commas (sometimes called **comma delimited** or **Comma Separated Value** files). Important: If you intend to use frozen Fibers (p. 1391), you must ensure that all release times are either be defined as 0 (zero) or the release field itself is omitted from the Custom Input definition file.
2. From the **Data** panel, click **Inlets and Outlets**, and the from the **Data Editors** panel, click the **Create Particle Custom Input** button. A new Custom Input component appears under **Inlets and Outlets** in the **Data** panel. Tip:You may also access this functionality from the right-click menu on the **Data** panel.
3. From the **Data** panel, click the Custom Input component you just added and then from the **Data Editors** panel, on the main **Particle Custom Input** tab, enter the **Name** you want, and then select the **Particle** set you want to use.
4. Click the **Load File** button.
5. From the **Select file to import** dialog, navigate to and select the definition file you saved in step 1, and then click **Open**. The table and fields under **File Info** on the **Data Editors** panel will display summary information from the file you imported.

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

6. If for the **Particle** parameter, you chose a multi-element (flexible and/or discrete-breakable) Custom Fiber (p. 1391) particle set that was defined using frozen Segments (p. 1394) (also known as a Frozen Fiber (p. 1391) shape; see also About Defining and Importing Custom Particle Shapes (p. 670)), and you want those frozen Segments to have motion assigned, select the motion you want applied from the **Motion Frame** list. Note: Unlike with custom geometries, particles assigned Motion Frames will not be visible in the **Motion Preview** window. (See also About Motion Preview Windows (p. 572).)

7. If you have enabled the **Thermal Model** for your simulation (see also About Physics Parameters (p. 417)) and the file you imported did not include a temperature column, you must also define what **Default Temperature** you want your particle set to have when released.

8. If you have enabled a module (p. 1392) for your simulation that requires a Particle Input parameter that was not defined in the custom injection file you imported, you must also define that parameter in the field provided on the **Particle Custom Input** tab.

See Also:

- About Adding and Editing Inlets and Outlets (p. 708)
- Create a New Particle Inlet (p. 740)
- Create a New Fluid Inlet (p. 741)
- Create a New Particle Custom Inlet (p. 743)
- Edit an Inlet or an Outlet (p. 745)
- Remove an Inlet or an Outlet (p. 745)
- About the Info Tab (p. 83)
- Use Colored Directional Handles (p. 126)

CREATE A NEW OUTLET

1. From the **Data** panel, click **Inlets and Outlets**, and the from the **Data Editors** panel, click the **Create Outlet** button. A new Outlet component appears under **Inlets and Outlets** in the **Data** panel. Tip:You may also access this functionality from the right-click menu on the **Data** panel.

2. From the **Data** panel, click the Outlet component you just added and then from the **Data Editors** panel, on the main **Outlet** tab, enter the **Name** and set the **Exit Point** you want.

3. From the **Particles** sub-tab, clear the **Enabled for Particles** if you do not want particles to be able to leave the simulation through this outlet. By default the checkbox is checked.

4. From the **SPH** sub-tab, do all of the following: a. Clear the **Enabled for Particles** if you do not want fluid to be able to leave the simulation through this outlet. By default the checkbox is checked. b. Select if you want to use a **Prescribed Pressure** boundary condition at the outlet. c. If the **Prescribed Pressure** option was selected, define the pressure that you want prescribed at this outlet.

See Also:

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

- About Adding and Editing Inlets and Outlets (p. 708)
- Edit an Inlet or an Outlet (p. 745)
- Remove an Inlet or an Outlet (p. 745)

EDIT AN INLET OR AN OUTLET

- From the **Data** panel, click the Inlet or Outlet you want to edit, and then from the **Data Editors** panel, modify the settings as desired.

See Also:

- About Adding and Editing Inlets and Outlets (p. 708)
- Create a New Particle Inlet (p. 740)
- Create a New Volumetric Inlet for Particles (p. 741)
- Remove an Inlet or an Outlet (p. 745)

REMOVE AN INLET OR AN OUTLET

- From the **Data** panel, click the Inlet or Outlet you want to remove, and then from the **Data Editors** panel, click the **Remove** button.

See Also:

- About Adding and Editing Inlets and Outlets (p. 708)
- Create a New Particle Inlet (p. 740)
- Create a New Volumetric Inlet for Particles (p. 741)
- Edit an Inlet or an Outlet (p. 745)

3.2.15. Collect (or Reject) Contacts Data

Whenever a particle collides (p. 1390) with another particle or boundary during a simulation, Rocky detects and calculates those interactions as one (or more) separate, individual contacts (p. 1390).

Rocky always calculates and makes use of both frictional and adhesive (if available) contacts data during simulation processing, but allows you to turn off the long-term collection of this data to reduce your files size.

You are able to post-process and visualize the contacts data that you choose to collect.

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

The collection of this data, as well as the post-processing and visualization, is done through a **Contacts** entity (p. 1395) that appears on the **Data** panel under **Study**.

Important:

Turning on Contacts data collection prior to processing is a requirement for post-processing and/or visualization of the data later.

What would you like to do?

- Learn more About Contacts (p. 746)
- Turn On Contacts Data Collection (p. 756)
- Turn Off Contacts Data Collection (p. 757)

See Also:

- Analyzing a Simulation (p. 931)

ABOUT CONTACTS

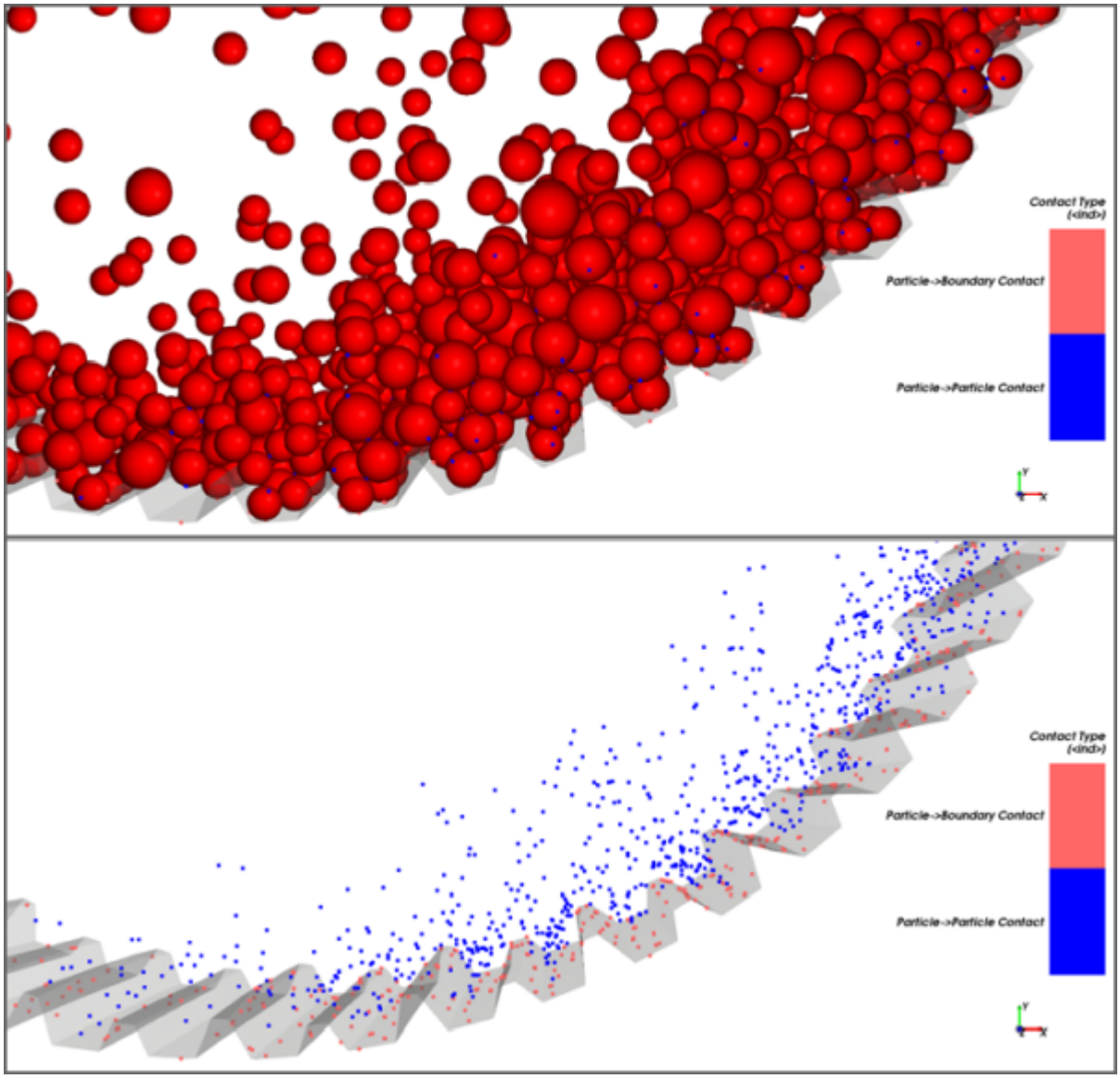
A contact (p. 1390) in Rocky refers to a specific location on a boundary or particle that has experienced a collision (p. 1390) with another particle during the simulation.

In addition to frictional contacts, which are the default type of contact calculated in Rocky, you can also collect adhesive contacts in this version. Adhesive contacts are present only if an **Adhesive Force** model is enabled in the simulation. (See also About Physics Parameters (p. 417).)

Rocky always calculates both frictional and adhesive contacts and makes use of that data during simulations. But you can also choose whether or not to keep one or both types of data after Rocky has used it. If you do choose to collect it, you can analyze and visualize that data (Figure 1) after processing, just like other kinds of data in Rocky. However, you must choose to collect it *before* processing a simulation.

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

Figure 3.178: Frictional contacts data being visualized with particles (top) and without particles (bottom)



Collect (or Reject) Contacts Data

The **Collect Contacts Data** checkbox, which is located on the **Data** panel's **Contacts** entity, is what determines whether or not the frictional-contact-related data required to calculate particle and boundary movement is saved for each output. The sub-checkbox **Include Adhesive Contacts** is what determines whether the adhesive-contact-related data is also saved. Note: You cannot save only adhesive contacts data. To save adhesive contacts data, you must also save frictional contacts data.

If you choose to enable one or both checkboxes prior to processing your simulation, your simulation file size will be larger but you will gain both of the following benefits:

- You will be able to analyze and visualize contacts data after processing.
- Resuming a Stopped simulation will be quicker as the contact data will be readily available.

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

When these checkboxes are cleared (default state), your simulation file size will be smaller but you will not be able to analyze nor visualize contacts data after processing and Resuming a Stopped simulation will take more time as the contact data for the in-process output time will need to be re-calculated before Processing is Resumed.

Tip:

View walk-through examples of collecting frictional contacts data in the following Workshops:

- Tutorial - SAG Mill in the *Rocky Tutorial Guide*
- Tutorial - Tablet Coater in the *Rocky Tutorial Guide*

Analyze and Visualize Contacts Data

If you have chosen to collect contacts data prior to processing, you will be able to post-process this data by using several new properties and user processes. See the sections below for more details.

STRESS-RELATED PROPERTIES FOR PARTICLES AND EULERIAN STATISTICS

A new property for Particles called **Stress Tensor** is available, plus several new stress-related properties for Eulerian Statistics.

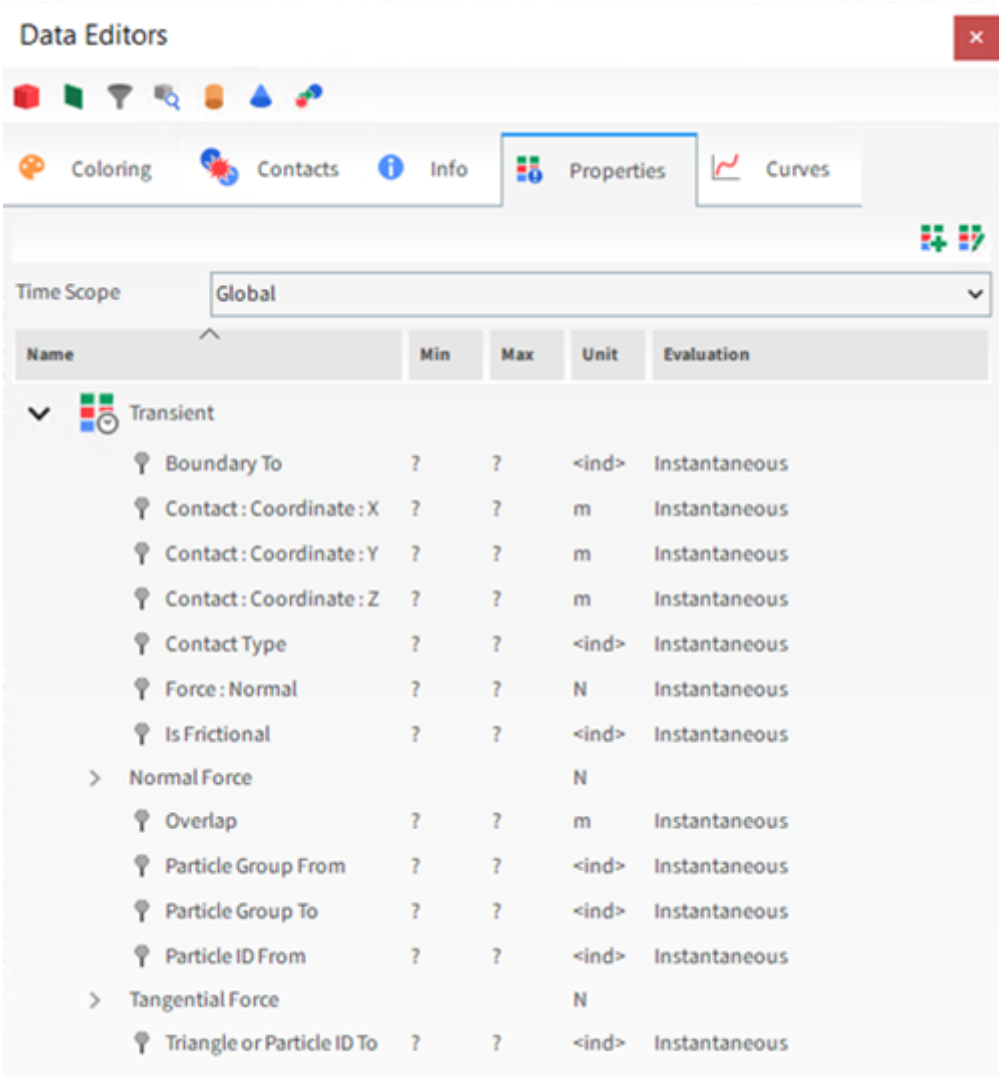
See also the definitions in the **Particles Properties (Simulation-Wide)** and the **Eulerian Statistics Properties** sections of the About Properties (p. 933) topic.

CONTACTS PROPERTIES

A new category of properties are available for the Contacts entity itself, enabling you to analyze co-ordinate locations, forces, and more for each contact.

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

Figure 3.179: Contacts Properties options on the Data Editors panel



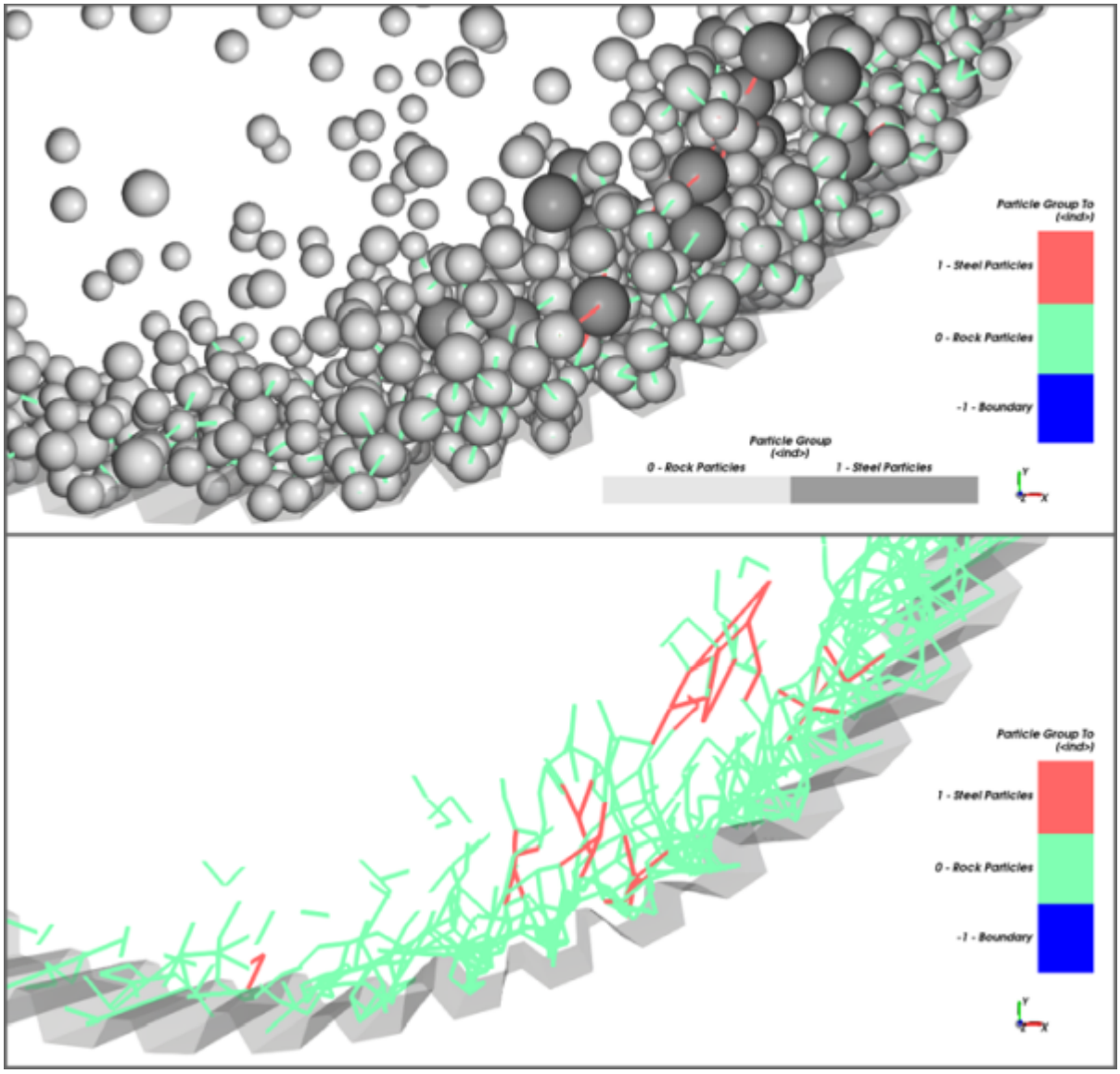
See also the **Contacts Properties** section of the About Properties (p. 933) topic.

VISUALIZE PARTICLE-TO-PARTICLE CONTACT NETWORKS

This functionality enables you to visualize 1D line segments connecting the centers of each particle that are interacting in a contact (Figure 3).

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

Figure 3.180: Particle-to-particle contact data visualized as line segments being shown with particles (top) and without particles (bottom)



This is accomplished by enabling the **Edges** option and then selecting the contact-related property you want to visualize.

Notes:

- Only particle-to-particle data can be visualized in a contact network; particle-to-boundary connections can not be visualized in this way—even if they are shown on the color scale. (See also About Color Scales (p. 1086).)
- Contact properties that can be visualized in a network are defined in the About Properties (p. 933) topic.
- Information about how to use the **Edges** option can be found in the About Using the Coloring Tab to Change a 3D View (p. 1074) topic.

CONTACTS USER PROCESSES

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

In addition to the new properties, there are two User Processes specifically designed for use with Contacts:

- **Particle to Contact**: This is designed to filter out which contacts are happening to a particular subset of particles. For example, you can:
  - Create a filter process of a particular particle group and then use the Particle to Contact User Process to analyze normal forces or overlaps from collisions involving the desired particle shape.
  - Create a filter process of particles within a given velocity range and then use the Particle to Contact User Process to analyze normal forces or overlaps from collisions involving them.

  This User Process is created from the main **Particles** entity or another User Process based upon it. Even though you create this User Process from Particles, its parent entity is **Contacts**. This means that after applying this User Process, you are able to further analyze the contacts by Contacts Properties.
- **Contact to Particle**: This is designed to filter out which particles are experiencing a particular subset of contacts. For example, you can:
  - Create a filter process of contacts affecting a particular boundary and then use the Contact to Particle User Process to analyze the particle properties-size distribution or size groups, for example- of these filtered contacts only.
  - Create a filter process of contacts with overlaps greater than a certain value, and then use the Contact to Particle User Process to analyze the PSD of those filtered particles. This can help you better understand why high overlaps are occurring with certain particles.

  This User Process is created from the main **Contacts** entity or another User Process based upon it. Even though you create this User Process from Contacts, its parent entity is **Particles**. This means that after applying this User Process, you are able to further analyze the particles by Particles Properties.

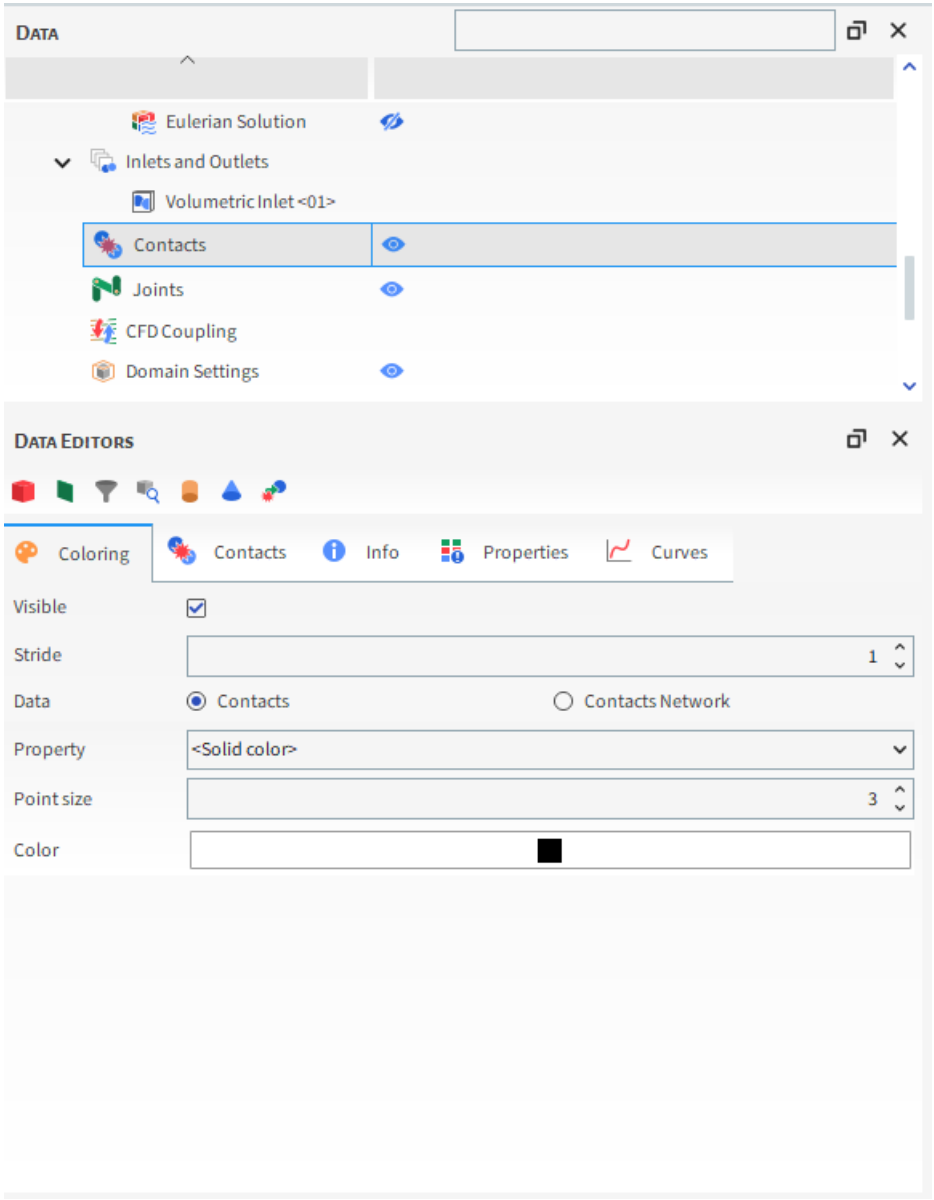
Important:

Even though both the **Contact to Particle** and the **Particle to Contact** User Processes will always be available, there will be no Contacts data to analyze unless you turn on the collection of Contacts data *prior to* processing your simulation. (See also Turn On Contacts Data Collection (p. 756).)

Contacts Parameter Definitions

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

Figure 3.181: Contacts parameters on the Data Editors panel (Coloring - Contacts)



Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

Figure 3.182: Contacts parameters on the Data Editors panel (Coloring - Contacts Network)

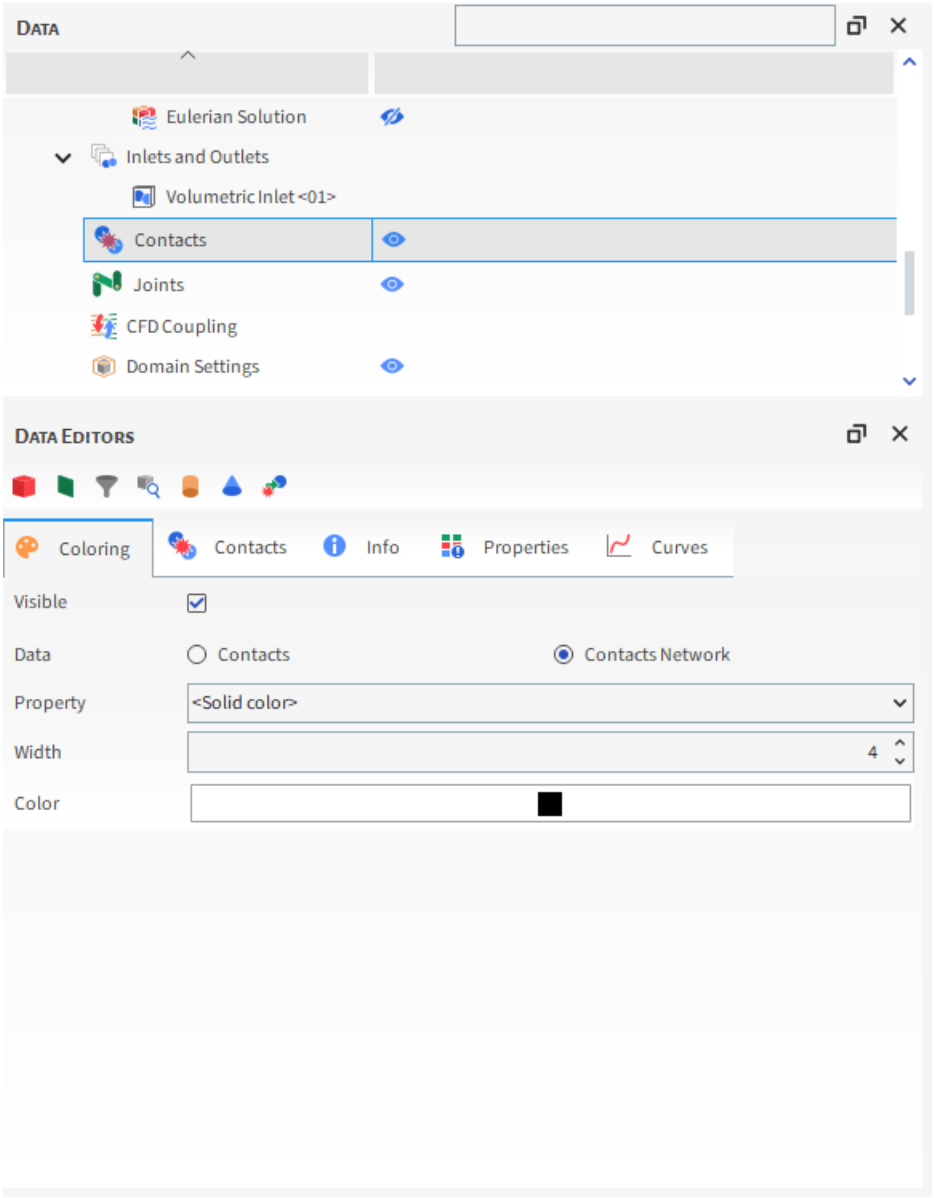


Table 3.4: Contacts Parameters - Coloring

Setting	Description	Range
Visible	When enabled, shows the selected entity in the active view window.	Turns on or off.
Stride	One out of this number of data points will be shown when displaying Contacts. The lower the number, the more data points will be displayed.	Whole numbers 1 or greater.
Data	**Contacts:** Enables the selected contacts to be colored in the 3D View.	Options limited by the selected Data.

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

Setting	Description	Range
	Contacts Network: Enables the selected connections between contacts to be colored in the 3D View.	
Property	Provide properties and other color options that apply to the display Data type within which the list is contained. • **<Solid Color>** applies a single color to the display Data type. • The remaining properties come directly from the component's Properties tab. Selecting one of these properties colors display Data type in a range according to the data collected for the current time step.	<Solid color>; List of properties automatically generated from the Properties tab.
Point size	When Contacts is selected, this changes the size of the dots used to draw the Contacts.	Positive Values.
Width	When Contacts Network is selected, this changes the thickness of Positive values the lines used to draw the borders.	Positive Values.
Color	When Contacts and Contacts Network are selected and the Property is **<Solid Color>**, this enables the selected display Data type to be colored in the one solid color chosen.	Options limited by the selected Color.

Use the figure and table below to help you understand the various Contacts parameters you can set for a simulation project.

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

Figure 3.183: Contacts parameters on the Data Editors panel

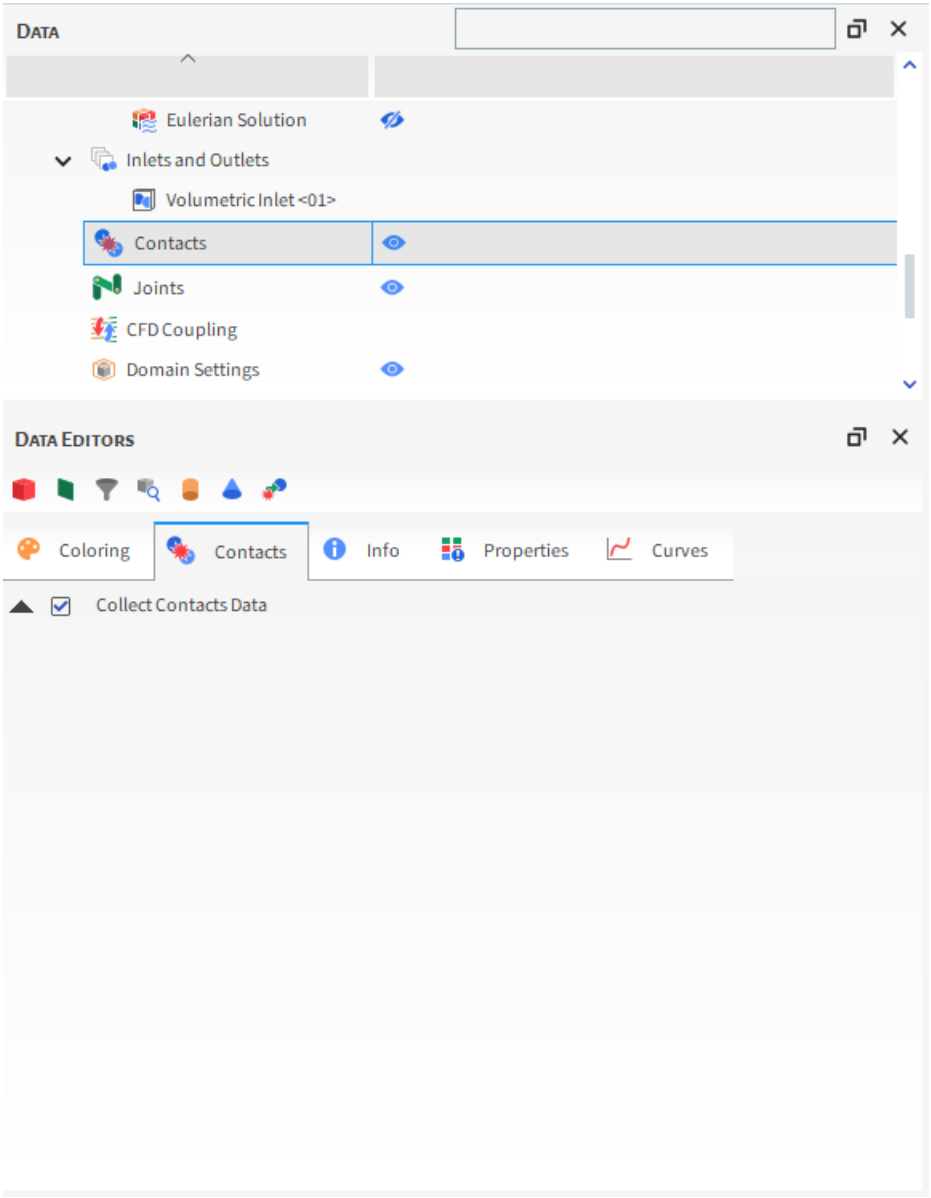


Table 1: Contacts parameter options

Setting	Description	Range
Collect Contacts Data	Determines whether or not the frictional-contacts- related data required to calculate particle and boundary movement is saved for each Output Time. When enabled, simulation file size will be larger but you will be able to visualize and analyze Contacts data later and Resuming a Stopped simulation will be quicker as the contact data will be readily available. When cleared, simulation file size will be smaller but you will not be able to visualize nor analyze Contacts data later and Resuming a Stopped simulation will take more time as the contact data for the in-process output will need to be recalculated before Processing is Resumed.	Turns on or off

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

Setting	Description	Range
	See also Resume Processing a Stopped Simulation.	
Include Adhesive Contacts	When an **Adhesive Force** is enabled, Rocky also saves for each Output Time the adhesive-contacts- related data required to calculate particle and boundary movement.	Turns on or off

What would you like to do?

- Turn On Contacts Data Collection (p. 756)
- Turn Off Contacts Data Collection (p. 757)
- Learn more Analyzing a Simulation (p. 931)
- Learn more about Analyzing Contacts from a Sub-Set of Particles (p. 1271)
- Learn more about Analyzing Particles from a Sub-Set of Contacts (p. 1274)

See Also:

- Resume Processing a Stopped Simulation (p. 913)
- About Properties (p. 933)
- About Solver Parameters (p. 464)
- About 3D View Windows (p. 1040)

TURN ON CONTACTS DATA COLLECTION

1. Set up your simulation as you normally would. (See also Setting Up a Simulation (p. 411).)
2. Prior to processing your simulation, do all of the following:
   a. From the **Data** panel, click **Contacts**.
   b. From the **Data Editors** panel, do one or both of the following:
      - Enable the **Collect Contacts Data** checkbox, which will save frictional contacts data.
      - (Optional) If you have enabled an **Adhesive Force** model, enable the **Include Adhesive Contacts** checkbox, which will also save adhesive contacts data. (See also About Physics Parameters (p. 417).)
3. Process your simulation as you normally would. (See also Processing a Simulation (p. 907).) The contacts data you chose to collect is available for post-processing.

See Also:

- About Contacts (p. 746)
- Turn Off Contacts Data Collection (p. 757)
- About Solver Parameters (p. 464)

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

TURN OFF CONTACTS DATA COLLECTION

In this version of Rocky, Contacts Data collection is turned off by default. However, if you have already turned on Contacts Data collection but now want to turn it off, you can use the following procedure.

1. Prior to processing your simulation, do all of the following:

   a. From the **Data** panel, click **Contacts**.

   b. From the **Data Editors** panel, ensure that both the **Include Adhesive Contacts** and the **Collect Contacts Data** checkboxes are cleared (unchecked).

2. Process your simulation as you normally would. (See also Processing a Simulation (p. 907).) The simulation file sizes are smaller and there are no Contacts data available for post-processing.

See Also:

- About Contacts (p. 746)
- Turn On Contacts Data Collection (p. 756)
- About Solver Parameters (p. 464)

3.2.16. Set or Modify Fluid and/or Air Flow Properties

In Rocky, there are several different methods for simulating the interaction between particles and the surrounding fluids (air, water, dust, etc.), known more commonly as Computational Fluid Dynamics (CFD). The CFD Coupling categories and methods available in Rocky are described below:

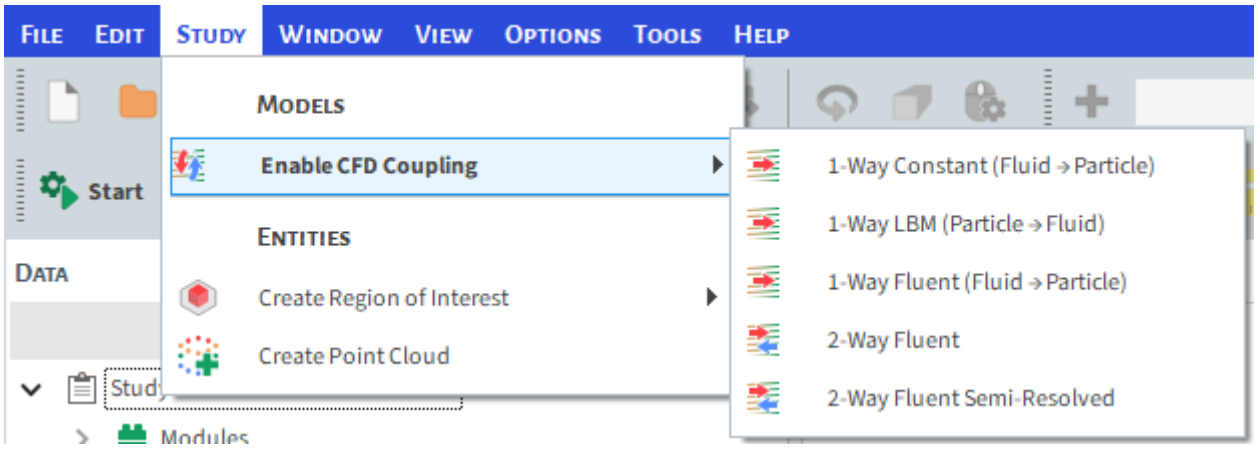
- **1-Way** | **Particle** → **Fluid**: In this method, particle flow affects the fluid flow but the fluid flow does not, in turn, affect the particles. In this, it is considered a one-way calculation and includes only the **LBM** option, which simulates how particles change the flow of fluids that are calculated using the Lattice Boltzmann Method (LBM).
- **1-Way** | **Fluid** → **Particle**: In these methods, the fluid flow affects the particle flow but the particle flow does not, in turn, affect the fluid flow. These are also considered one-way calculations and include the following two methods:
  - The **Constant** method, which simulates how a constant fluid flow changes the flow of particles in Rocky.
  - The **Fluent** method, which simulates how the steady state or transient fluid flow calculated in Ansys Fluent changes the flow of the particles with which they come into contact in Rocky.
- **2-Way** | **Fluid [left-right arrow] Particle**: In these methods, the fluid flow affects the particle flow and vice versa, resulting in a truly two-way coupled simulation. This version of Rocky includes the following methods:
  - The **Fluent** method, which considered an unresolved method and is appropriate for most 2-Way coupling cases.
  - The **Fluent Semi-Resolved** method, which is a special 2-Way coupling method most appropriate for a few, very large (when compared to the cell size of the Fluent mesh) particles interacting with a dynamic fluid whose accurate behavior requires a very refined Fluent mesh. It should be

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

used in cases where the resolution of the fluid flow around the particles is critical to capturing the behavior of the particles, and relying on standard correlations for fluid forces based on the centroid fluid velocity and velocity gradients isn't sufficient.

To select the desired CFD coupling method, go to the top menu and navigate to **| Study | Enable CFD Coupling**, then choose the appropriate method.

Figure 3.184: Dynamic placement of CFD coupling



You may choose only one CFD Coupling method per simulation project file. However, for most of these methods, you can choose how the data is visualized in the Coloring tab. (See also Create and Modify a 3D View (p. 1040).)

See further details in the sections below.

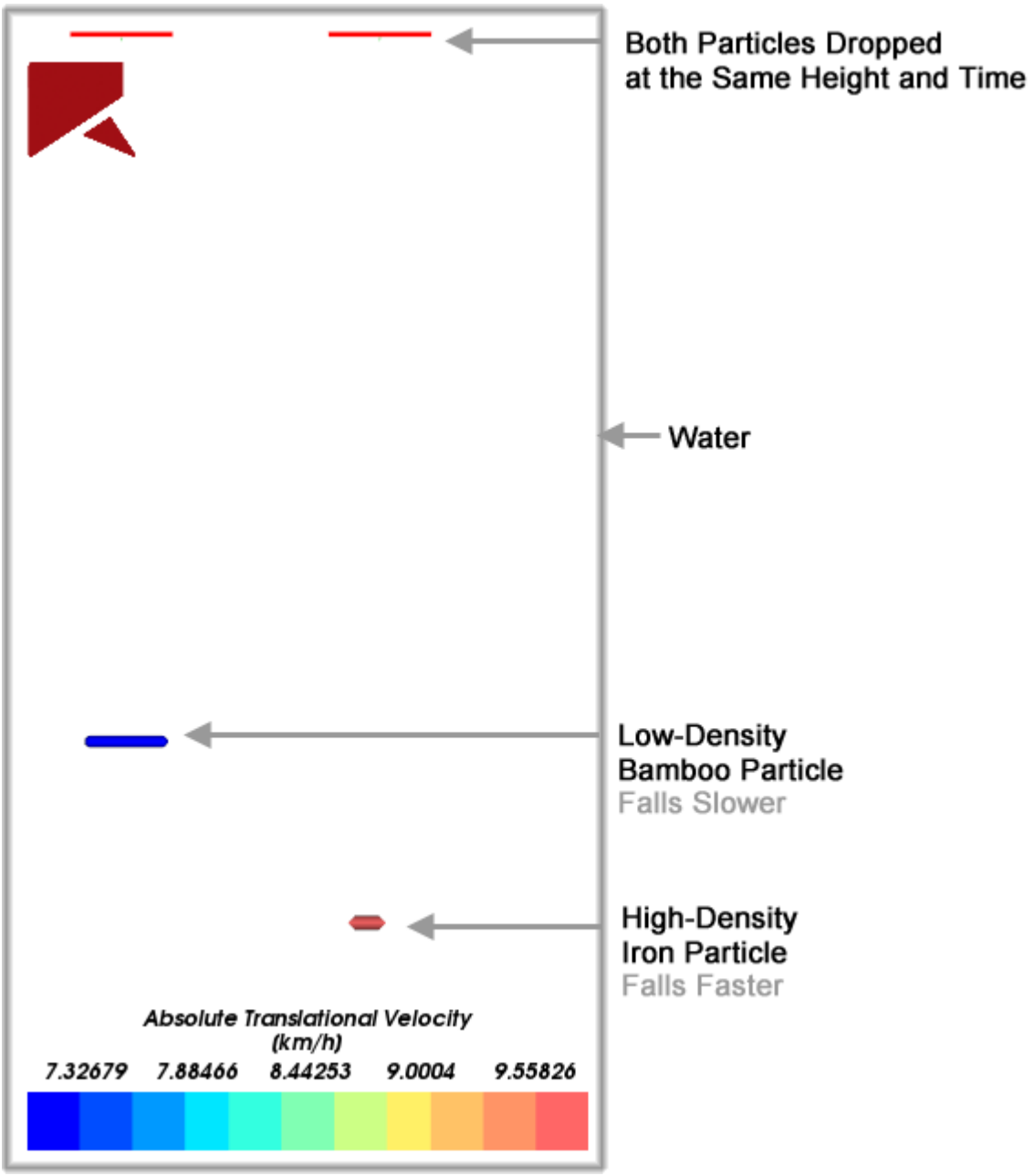
1-WAY CONSTANT METHOD

This method is useful in cases where you have a known, unchanging fluid flow and want to understand how it impacts the flow of particles without having to use a separate CFD program. You can set a constant value for density, velocity, viscosity (and thermal properties if thermal is being solved), and Rocky will use those values to simulate the fluid flow.

For example, with this method you can simulate the relative buoyancy of a dense and light material through water (Figure 2), or simulate the relative speed of a stretched and compact shape through air.

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

Figure 3.185: 1-Way Constant method showing shapes of different densities falling through water



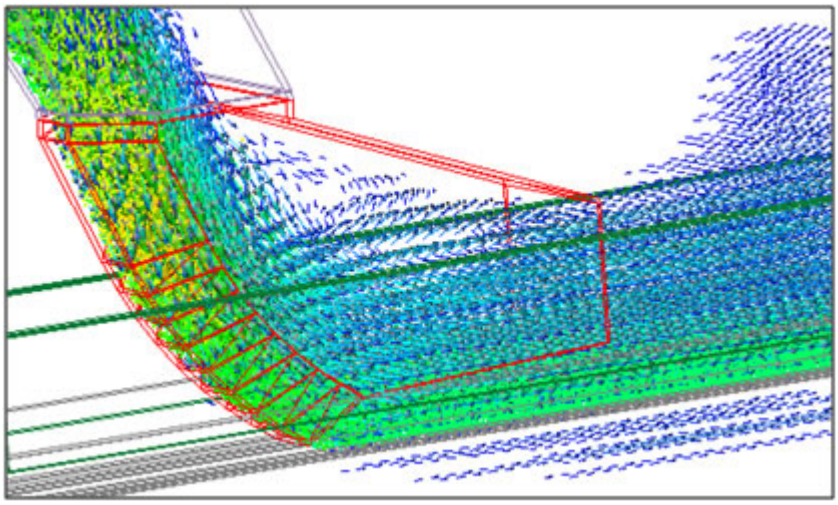
With this method, the particle flow does not affect the fluid flow.

1-WAY LBM METHOD

This method is accomplished by enabling Rocky to calculate how particle flow and the boundaries that contain those particles affects the air that comes into contact with them. This method uses the Lattice Boltzmann Method (LBM) for its calculations and is useful for simulating how much dust a transfer chute design would generate, for example. In this method, the air flow does not affect the movement of the particles; only the particles and the boundaries affect the air flow. (See Figure 1 for an example.)

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

Figure 3.186: 1-Way LBM method showing how dust-like vectors interact with the particles



1-WAY FLUENT METHOD

This method is accomplished by a one-way coupling with Ansys Fluent-using either steady state or transient data.

In this method, a CFD simulation done within Fluent calculates the velocity and pressure fields generated by the fluid as it flows through the equipment being studied. Once that Fluent case is complete, you can choose one of two data export methods within Fluent:

- Export a single file of the fluid data at steady state.
- Record and then export a set of transient fluid data.

After you export this data out of Fluent, you import it into Rocky. During the simulation, Rocky will then calculate how the fluid flow would affect the particle flow.

This method is particularly useful for simulating the effect of water on the movement of particles through a pipe, for example. Or, for simulating the transport of particles with different densities by water in a slurry-like flow. In the 1-Way Fluent method, the particles do not affect the fluid flow; the fluid flow affects the movement of the particles. (See Figure 3 for an example.)

While the fluid data must still come from Fluent, with the 1-Way Fluent method it is not necessary to have both Rocky and Fluent installed on the same machine.

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

Figure 3.187: 1-Way Fluent with steady state fluid data showing how particles interact with the fluid-like vectors



2-WAY FLUENT METHOD

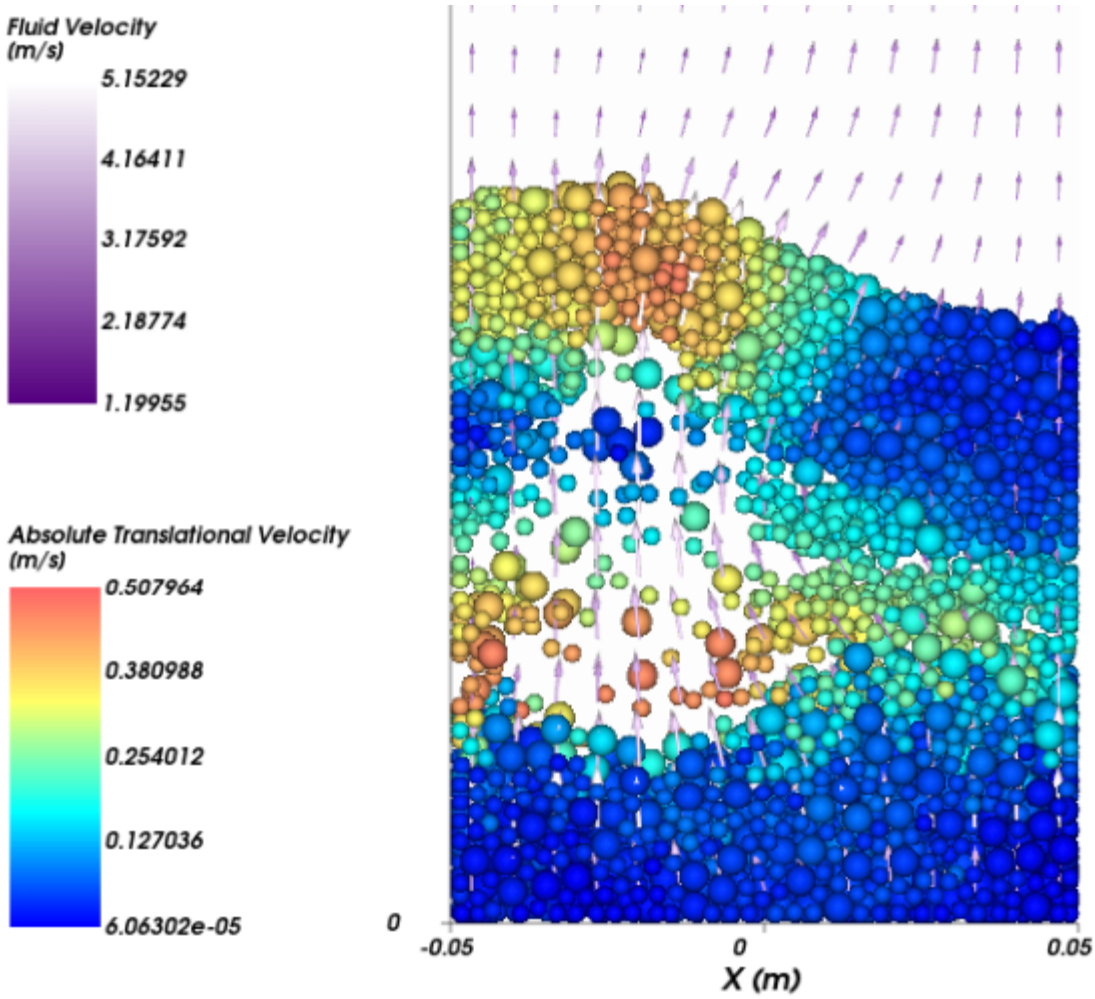
This unresolved method is particularly useful for simulating complex phenomena such as pneumatic conveying, granular drying, slurry flow inside grinding mills, or even chemical reactions between particles and fluids. In the 2-Way Fluent method, the particles are part of the fluid flow and will affect it in a two-way interaction: particles are affected by other particles and the fluid around it while the fluid flow is also affected by the particle pressure. (See Figure 4 for an example.)

In this method, correlations for fluid forces acting on particles (such as drag, lift, and virtual mass) are calculated based on the fluid quantities (such as velocity and velocity gradient) of the cell where the centroid of the particle is located. This flow approximation means that your mesh can be larger than with the 2-Way Fluent Semi-Resolved method.

Unlike the 1-Way Fluent Method, for the 2-Way Fluent method to work, both Fluent and Rocky must be installed on the same host machine.

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

Figure 3.188: 2-Way Fluent method showing colored particles reacting to a fluidized bed



2-WAY FLUENT SEMI-RESOLVED METHOD

This semi-resolved method is useful in cases where the resolution of the fluid flow around the particles is critical to capturing the behavior of the particles, and relying on standard correlations for fluid forces based on the centroid fluid velocity and velocity gradients-as done in the standard 2-Way Fluent method-isn't sufficient. Rather than solving correlations based upon the centroid velocities, fluid forces are integrated at the particle's surface in the semi-resolved method, which requires a very fine mesh around the particle in order to work properly.

This method therefore requires a certain number of cells to define the particle surface, which means a particle-grid size ratio greater than 10. The finer the mesh resolution, the more accurate the results (but also, the more computationally expensive the simulation).

This method works with both single-phase and multiphase flows, including support for the VOF model in Fluent, and accepts spheres, shells (both rigid and flexible), and custom solid shapes (rigid only).

Like the standard 2-Way Fluent method, the 2-Way Fluent Semi-Resolved method requires that both Fluent and Rocky be installed on the same host machine.

What would you like to do?

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

- Use the 1-Way Constant Method (p. 763)
- Use the 1-Way LBM Method (p. 770)
- Use the 1-Way Fluent (p. 775)
- Use the 2-Way Fluent Method (p. 795)
- Use the 2-Way Fluent Semi-Resolved Method (p. 830)

See Also:

- Set Simulation Parameters (p. 414)

USE THE 1-WAY CONSTANT METHOD

The 1-Way Constant CFD Coupling method simulates how a constant fluid flow changes the flow of particles in Rocky without the need for a separate CFD Program.

What would you like to do?

- Learn more About Using the 1-Way Constant Method (p. 763)
- Simulate a Constant Fluid Flow Using Only Rocky (p. 770)

See Also:

- Set or Modify Fluid and/or Air Flow Properties (p. 757)
- Use the 1-Way LBM Method (p. 770)
- Use the 1-Way Fluent Method (p. 775)
- Use the 2-Way Fluent Method (p. 795)
- Set Simulation Parameters (p. 414)

About Using the 1-Way Constant Method

The 1-Way Constant CFD Coupling method uses the known, unchanging fluid values you provide to simulate a constant fluid flow in Rocky, without the need for a separate CFD Program. Because the fluid flow stays constant throughout the whole simulation, it is somewhat more limiting when compared to the 1-Way Fluent (p. 775), but can still be quite useful in certain situations. For example, when simulating the fluid's effect on particles as objects are dropped through water or air.

The 1-Way Constant method is also compatible with Thermal Modeling (p. 400), so you can also simulate how the temperature of the fluid affects the particles that come into contact with it.

LIMITATIONS

When combined with Coarse Grain models, only drag forces are considered. This means that other non-drag fluid force models, such as virtual mass and lift, are incompatible with Coarse Grain modeling. (See also About Physics Parameters (p. 417).)

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

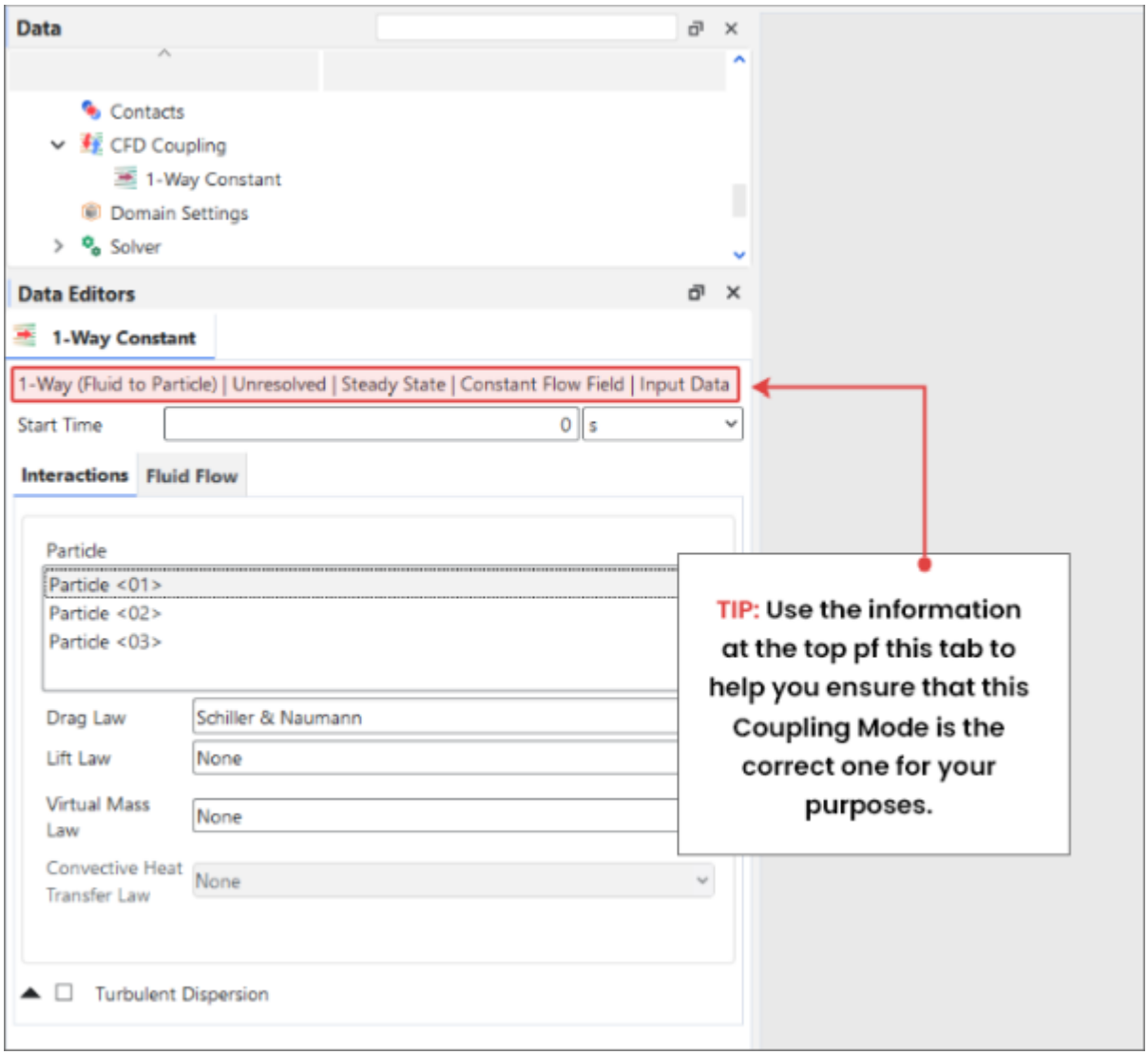
In addition, Shell (p. 1395) and Solid (p. 1395) particles composed of Multiple Elements (also known as meshed (p. 1392) or flexible particles) are incompatible with 1-Way Constant coupling. The only flexible particle shapes compatible with this feature are Fibers (p. 1391). (See also About Adding and Editing Particle Sets (p. 602).)

For a full list of limitations, refer to the Ansys and CFD Coupling Limitations (p. 63) topic.

1-WAY CONSTANT PARAMETER DEFINITIONS

Use the figures and table below to understand the various parameters you can set for 1-Way Constant.

Figure 3.189: CFD Coupling, 1-Way Constant in the Data Editors panel



Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

Figure 3.190: CFD Coupling, 1-Way Constant, Fluid Flow Tab in the Data Editors panel

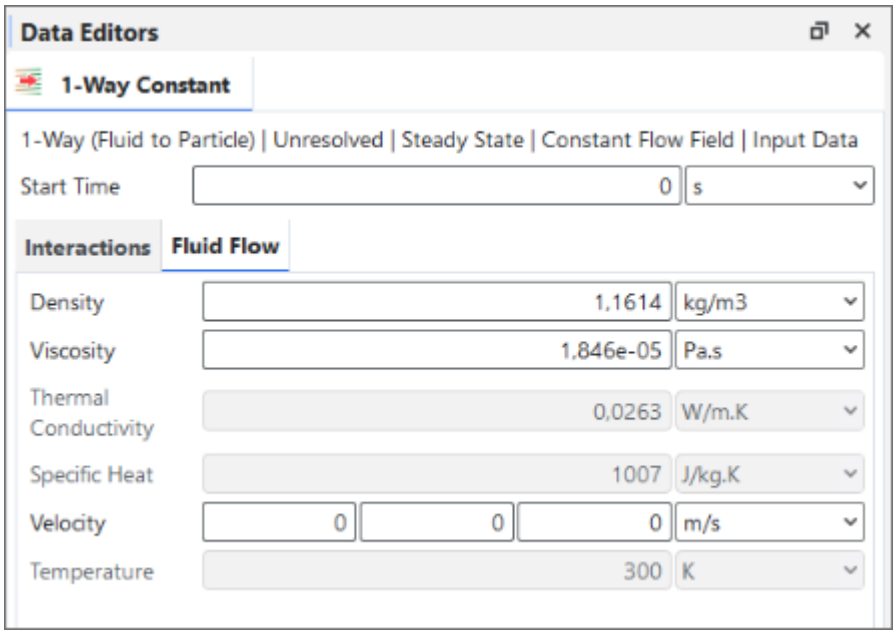


Table 1: 1-Way Constant Settings (all tabs)

Setting	Description	Range
Start Time	The time during the simulation when you want to start taking fluid flow into account.	Positive values
Interactions tab		
Particle	Permits correlations- such as drag, lift, virtual mass, and convective heat transfer laws (if applicable)- to be defined per *Particle set* (p. 1393). You accomplish this by selecting (or multi-selecting) from a list of previously defined Particle sets, and then defining the applicable CFD Laws for only the selected set (or sets). **Important:** Even though they will still be listed here, neither flexible *Shell* (p. 1395) nor flexible *Solid* (p. 1395) particle shapes are supported in CFD Coupling approaches, including this one. (See also Ansys and CFD Coupling Limitations (p. 63)).	Automatically provided
Drag Law	Correlation used by Rocky to calculate drag forces on particles (with the exception of *Fiber* (p. 1391) and also *Solid* (p. 1395) Sphero-Cylinder shapes-see note below):	Schiller & Naumann; Haider & Levenspiel; Ganser; Dallavalle: Marheineke & Wegener; Morsi & Alexander

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

Setting	Description	Range
	• **Schiller & Naumann**: Recommended for Spheres only. • **Haider & Levenspiel**: Recommended for Spheres and isometric *shaped particle* (p. 1395) since it takes the particle sphericity into account in order to calculate the drag coefficient. • **Ganser**: Recommended for both spherical and shaped particles since it considers the effect of shape and alignment of the particle with the flow when computing the drag coefficient. Note: Ganser is incompatible with *Assembly* (p. 1389), *concave* (p. 1390), and *Shell* (p. 1395) particle shapes. (See also About Adding and Editing Particle Sets (p. 602)). • **Dallavalle**: Recommended for Spheres only. Although it yields results similar to the **Schiller & Naumann** drag law, as a continuous function, it can lead to more stable simulations on very sensitive cases. • **Marheineke & Wegener**: The *only* drag law currently recommended for use with *Fiber* (p. 1391) and also *Solid* (p. 1395) Sphero-Cylinder particle shapes. • **Morsi & Alexander**: Recommended for Spheres, but can be adapted to shaped particles when the adjustable coefficients are used. Best used in cases where there is a desire to tune the drag law to match experimental results. **Note:** Unless stated otherwise, the above drag laws apply to any supported particle shapes (see also Ansys and CFD Coupling Limitations (p. 63)) except *Fiber* (p. 1391) and also *Solid* (p. 1395) Sphero-Cylinder shapes, which have their own modified drag law based on the work by	Note: If you have an *external Module* (p. 1391) that adds in an additional CFD law, you must use that law at least once in your simulation project if the Module is enabled. Refer to the Module's documentation (if provided) for more information. (See also Rocky Simulation Entities that can be Affected by Modules (p. 223)).

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

Setting	Description	Range
	Marheineke and Wegener that should always be used for those shapes. **Tip:** For more information about these laws, refer to the CFD Coupling Technical Manual. (From the Rocky **Help** menu, point to **Manuals**, and then click **CFD Coupling Technical Manual**.)	
Lift Law	Correlation used by Rocky to calculate the fluid-particle force transversal to the direction of the relative velocity.	None; Saffman; Mei Note: If you have an *external Module* that adds in an additional CFD law, you must use that law at least once in your simulation project if the Module is enabled. Refer to the Module's documentation (if provided) for more information. (See also Rocky Simulation Entities that can be Affected by Modules (p. 223)).
Virtual Mass Law	Correlation used by Rocky to calculate the virtual mass coefficient.	None; Constant Note: If you have an *external Module* that adds in an additional CFD law, you must use that law at least once in your simulation project if the Module is enabled. Refer to the Module's documentation (if provided) for more information. (See also Rocky Simulation Entities that can be Affected by Modules (p. 223)).
Convective Heat	When **Thermal Model** is enabled (see also About Physics Parameters (p. 417)) this correlation is used	None; Ranz & Marshall; Whitaker

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

Setting	Description	Range
Transfer Law	by Rocky to calculate the fluid-particle convective heat transfer coefficient.	Note: If you have an *external Module* that adds in an additional CFD law, you must use that law at least once in your simulation project if the Module is enabled. Refer to the Module's documentation (if provided) for more information. (See also Rocky Simulation Entities that can be Affected by Modules (p. 223)).
Morsi & Alexander Parameters Use Defined Constants	When **Morsi & Alexander** is selected for **Drag Law**, this enables you to determine how the model's three coefficients will be determined. Specifically: • When cleared, the standard values provided with the model will be used. See the *CFD Coupling Technical Manual* for more detail. (From the Rocky **Help** menu, point to **Manuals**, and then click *CFD Coupling Technical Manual*.) • When enabled, you can define your own set of coefficients that will be used by the model. Tip: Defining your own constants can be particularly useful in cases where you have experimental data measuring the drag of fluid over particles that you want matched in the simulation by adjusting the coefficients of the drag law.	Turns on or off
K_1	When **Use Defined Constants** is enabled, this is the value you want assigned to the K_1 constant used in the **Morsi & Alexander** drag law calculation. See also the *CFD Coupling Technical Manual*. (From the Rocky **Help** menu, point to **Manuals**, and then click *CFD Coupling Technical Manual*.)	Any value
K_2	When **Use Defined Constants** is enabled, this is the value you want assigned to the K_2 constant used in the **Morsi & Alexander** drag law calculation. See also the *CFD Coupling Technical Manual*. (From the Rocky **Help** menu, point to **Manuals**, and then click *CFD Coupling Technical Manual*.)	Any value
K_3	When **Use Defined Constants** is enabled, this is the value you want assigned to the K_3 constant used in the **Morsi & Alexander** drag law calculation. See also the *CFD Coupling Technical Manual*. (From the	Any value

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

Setting	Description	Range
	Rocky **Help** menu, point to **Manuals**, and then click *CFD Coupling Technical Manual*.)	
Turbulent Dispersion	Determines whether or not the dispersion of particles due to turbulence will be accounted for. Specifically: • When cleared, turbulent dispersion will not be modeled. • When enabled, the Discrete Random Walk Model will be used to calculate the dispersion of particles due to turbulence.	Turns on or off
Turbulent Kinetic Energy	When **Turbulent Dispersion** is enabled, this is the amount of (constant) turbulent kinetic energy that will be used when modeling particle dispersion due to turbulence.	Positive values
Turbulent Dissipation Rate	When **Turbulent Dispersion** is enabled, this is the (constant) turbulent kinetic energy dissipation rate that will be used when modeling particle dispersion due to turbulence.	Positive values
Buoyancy	Determines whether or not hydrostatic pressure, which is used for buoyancy calculations, is included for the fluid-particle interactions. **Note:** This setting applies to all fluid-particle interactions regardless of what is selected for **Particle**.	Turns on or off
Fluid Flow tab		
Density	The constant density of the fluid.	Positive values
Viscosity	The constant viscosity of the fluid.	Positive values
Thermal Conductivity	When **Thermal Model** is enabled (see also About Physics Parameters (p. 417)) this is the constant thermal conductivity of the fluid.	Any value
Specific Heat	When **Thermal Model** is enabled (see also About Physics Parameters (p. 417)) this is the constant specific heat value of the fluid.	Any value
Velocity	The constant velocity vector of the fluid, the magnitude of which is made up of the x, y, and z components respectively.	Any value

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

Setting	Description	Range
Temperature	When **Thermal Model** is enabled (see also About Physics Parameters (p. 417)) this is the constant temperature of the fluid.	Values above but not equal to 0 K (zero Kelvin)

What would you like to do?

- Simulate a Constant Fluid Flow Using Only Rocky (p. 770)

See Also:

- Set or Modify Fluid and/or Air Flow Properties (p. 757)
- I cannot find Particle Size Scale Factor (p. 1330)

Simulate a Constant Fluid Flow Using Only Rocky

1. If you want to model thermal properties in your simulation, ensure you enable the Thermal Model during the Set Simulation-Wide Parameters (p. 414) step. (See also Enable Thermal Modeling Calculations (p. 400).)
2. From the **Data** panel, click **CFD Coupling** and then from the **Data Editors** panel, choose **Constant (Fluid → Particle)** from the **Coupling Mode** list. The **1-Way Constant** component appears in the **Data** panel beneath **CFD Coupling**.
3. In the **Data** panel, click the new **1-Way Constant** component.
4. From the **Data Editors** panel, on the **1-Way Constant** tab, do the following:
    a. Set the **Start Time** you want.
    b. On the **Interactions** tab, do the following:
        i. From the **Particle** list, select (or multi-select) the Particle set name(s) for which you want to define particle-fluid interactions, and then select the various CFD laws you want for the selected set(s).
        ii. Select what you want for **Turbulent Dispersion** and **Buoyancy**.
    c. On the **Fluid Flow** tab, define the constant **Density**, **Viscosity**, **Velocity**, and other thermal-related values you want.

The fluid flow data will be calculated when you process the simulation. (See also Processing a Simulation (p. 907).)

See Also:

- About Using the 1-Way Constant Method (p. 763)
- I cannot find the Torque Law setting for 1-Way Constant (p. 1336)

USE THE 1-WAY LBM METHOD

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

This type of CFD Coupling uses the Lattice Boltzmann Method (LBM) to simulate how the density, viscosity, and size of the air flow cells is affected by the particle flow.

What would you like to do?

- Learn more About Using the 1-Way LBM Method (p. 771)
- Turn On 1-Way LBM Properties (p. 774)
- Modify 1-Way LBM Properties (p. 774)
- Turn Off 1-Way LBM Properties (p. 774)
- Learn more about CFD Coupling Limitations (p. 63)

See Also:

- Set or Modify Fluid and/or Air Flow Properties (p. 757)
- Use the 1-Way Constant Method (p. 763)
- Use the 1-Way Fluent Method (p. 775)
- Use the 2-Way Fluent Method (p. 795)
- Set Simulation Parameters (p. 414)

About Using the 1-Way LBM Method

This type of CFD Coupling uses the Lattice Boltzmann Method (LBM) to simulate how air and dust flow are affected by the flow of particles in the simulation. Because of this, air flow changes as the simulation progresses. You need no special files to conduct this kind of analysis but note that the extra calculations will slow down the simulation processing.

At any point, you can see how your 1-Way LBM simulation results look in a 3D View (see also Create and Modify a 3D View (p. 1040)).

Tip:

For a self-guided walk-through example of setting up and running a simulation using the Lattice Boltzmann (LBM) Air Flow Method, refer to Tutorial - Discharge Air Flow in the *Rocky Tutorial Guide*.

LIMITATIONS

The 1-Way LBM CFD Coupling method is incompatible with Coarse Grain Modeling. (See also About Physics Parameters (p. 417).)

It is also incompatible with Shell (p. 1395) and Solid (p. 1395) particles composed of Multiple Elements (also known as meshed (p. 1392) or flexible particles). The only flexible particle shapes compatible with this feature are Fibers (p. 1391). (See also About Adding and Editing Particle Sets (p. 602).)

For a full list of limitations, refer to the Ansys and CFD Coupling Limitations (p. 63) topic.

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

1-WAY LBM PARAMETER DEFINITIONS

Use the figure and table below to understand the various parameters you can set for 1-Way LBM.

Figure 3.191: CFD Coupling, 1-Way LBM tab in the Data Editors panel

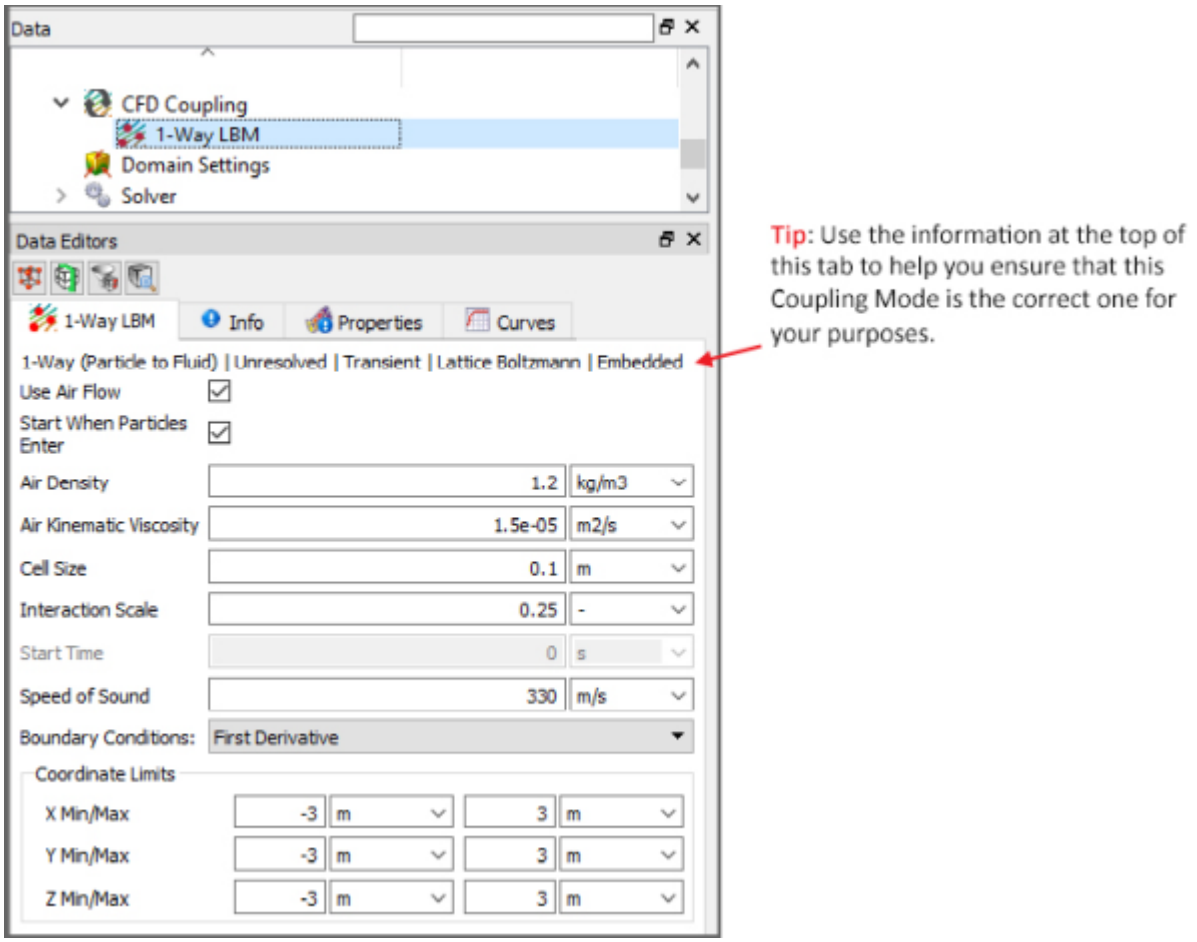


Table 1: 1-Way LBM Settings

Setting	Description	Range
Use Air Flow	Enables Lattice Boltzmann Air Flow calculations to be included in the simulation.	Turns on or off
Start When Particles Enter	Enables Lattice Boltzmann Air Flow calculations to start as soon as particles appear in the air flow limit box.	Turns on or off
Air Density	The density of the air.	Positive values
Air Kinematic Viscosity	The kinematic viscosity of the air.	Positive values
Cell Size	The size of an air flow cell.	Positive values
Interaction Scale	Determines the interaction between the particle and the air.	Positive values. (It is recommended that the default

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

Setting	Description	Range
		value of 0.25 is used.)
Start Time	When **Start When Particles Enter** is cleared, this is the time that Lattice Boltzmann Air Flow calculations begin during the simulation.	Positive values
Speed of Sound	The speed of sound, which is used to calculate air flow.	Positive values
Boundary Conditions		
First Derivative	This order provides more stability to the calculations but less accuracy. **Tip:** For more information on how Rocky calculates boundary conditions, see the 2000 paper by Verberg and Ladd entitled, "Lattice-Boltzmann Model with Sub-Grid-Scale Boundary Conditions."	Turns off or on
Second Derivative	This order provides more accuracy to the calculations but less stability.	Turns off or on
Coordinate Limits		
X Min/Max	Distance away from zero on the X axis to place the nearest and farthest corner of the air flow limit box.	No limit
Y Min/Max	Distance away from zero on the Y axis to place the nearest and farthest corners of the air flow limit box.	No limit
Z Min/Max	Distance away from zero on the Z axis to place the nearest and farthest corners of the air flow limit box.	No limit

What would you like to do?

- Turn On 1-Way LBM Properties (p. 774)
- Modify 1-Way LBM Properties (p. 774)
- Turn Off 1-Way LBM Properties (p. 774)
- Learn more about CFD Coupling Limitations (p. 63)

See Also:

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

- Set or Modify Fluid and/or Air Flow Properties (p. 757)
- Set Simulation Parameters (p. 414)
- My "Air Viscosity" units changed (p. 1334)

Turn On 1-Way LBM Properties

1. From the **Data** panel, click **CFD Coupling**.
2. From the **Data Editors** panel, choose **LBM (Particle → Fluid)** from the **Coupling Mode** list. The **1-Way LBM** component appears in the **Data** panel beneath **CFD Coupling**.
3. In the **Data** panel, click the new **1-Way LBM** component.
4. From the **Data Editors** panel, on the **1-Way LBM** tab, ensure the **Use Air Flow** checkbox is enabled, and then modify the settings as you want. The LBM air flow data will be calculated when you process the simulation. (See also Processing a Simulation (p. 907).)

See Also:

- About Using the 1-Way LBM Method (p. 771)
- Modify 1-Way LBM Properties (p. 774)
- Turn Off 1-Way LBM Properties (p. 774)

Modify 1-Way LBM Properties

1. From the **Data** panel, under **CFD Coupling**, click **LBM (Particle → Fluid)**.
2. From the **Data Editors** panel, on the **1-Way LBM** tab, modify the settings as you want.

See Also:

- About Using the 1-Way LBM Method (p. 771)
- Turn On 1-Way LBM Properties (p. 774)
- Turn Off 1-Way LBM Properties (p. 774)

Turn Off 1-Way LBM Properties

1. From the **Data** panel, under **CFD Coupling**, click **LBM (Particle → Fluid)**.
2. From the **Data Editors** panel, on the **1-Way LBM** tab, clear the **Use Air Flow** check box.

Tip:

You can also remove 1-Way LBM from your set up completely by choosing **No Coupling** from the **Coupling Mode** list. (From the **Data** panel, click **CFD Coupling** and then from the **Data Editors** panel, make your selection on the **CFD Coupling** tab.)

See Also:

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

- About Using the 1-Way LBM Method (p. 771)
- Turn On 1-Way LBM Properties (p. 774)
- Modify 1-Way LBM Properties (p. 774)

USE THE 1-WAY FLUENT METHOD

The 1-Way Fluent method provides an interface to generate a set of fluid flow data files in Ansys Fluent—-which can include air, water, and other fluids—-that you can then import into Rocky. In this version of Rocky, the fluid data you export can represent a steady state or transient flow.

This process can be conducted with Rocky and Fluent as standalone products, or through Ansys Workbench.

Important:

The *Fluent GPU* solver is not available in the Ansys Workbench enviroment (see the Fluent in Workbench User's Guide). Consequently, if you want to use the *Fluent GPU* solver, you should export the data files using the Fluent standalone version.

What would you like to do?

- Learn more About Using the 1-Way Fluent Method (p. 775)
- Import Fluid Flow Data from Ansys Fluent (p. 788)
- Learn more About Rocky and Ansys Workbench Integration (p. 144)
- Set Up and Run a 1-Way Fluent Transient Project within Ansys Workbench (p. 789)
- Install Ansys Coupling Components (p. 792)
- Uninstall Ansys Coupling Components (p. 795)

See Also:

- Set or Modify Fluid and/or Air Flow Properties (p. 757)
- Use the 1-Way LBM Method (p. 770)
- Use the 1-Way Constant Method (p. 763)
- Use the 2-Way Fluent Method (p. 795)
- Set Simulation Parameters (p. 414)

About Using the 1-Way Fluent Method

The 1-Way Fluent Method is a CFD Coupling option that enables Rocky to receive fluid flow data from Ansys Fluent-at either a steady or transient state—and then calculate how the particles react to that flow.

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

It is called a 1-Way Coupling simulation because Rocky receives data only once from Fluent and sends nothing to Fluent in return.

Tip:

You may also set up and process a 1-Way Fluent simulation within Ansys Workbench. To accomplish this with transient Fluent data, however, there are separate steps you must follow. (See also Set Up and Run a 1-Way Fluent Transient Project within Ansys Workbench (p. 789).)

SPECIAL REQUIREMENTS FOR 1-WAY FLUENT COUPLING

To make it possible to share the Fluent fluid flow data with Rocky, Rocky includes a special **Ansys Fluent Coupling Support** component. Installing this component enables a new **Rocky Export** menu option in Fluent (Figure 3.192: Rocky Export menu in Fluent (p. 776)) that allows the fluid data to be packaged into a unique "Fluent to Rocky" or F2R file, that later gets imported into Rocky as part of the 1-Way Fluent coupling process.

Important:

The 1-way Fluent method (**Rocky Export** menu) is not available in Fluent CFD Pro license.

Figure 3.192: Rocky Export menu in Fluent

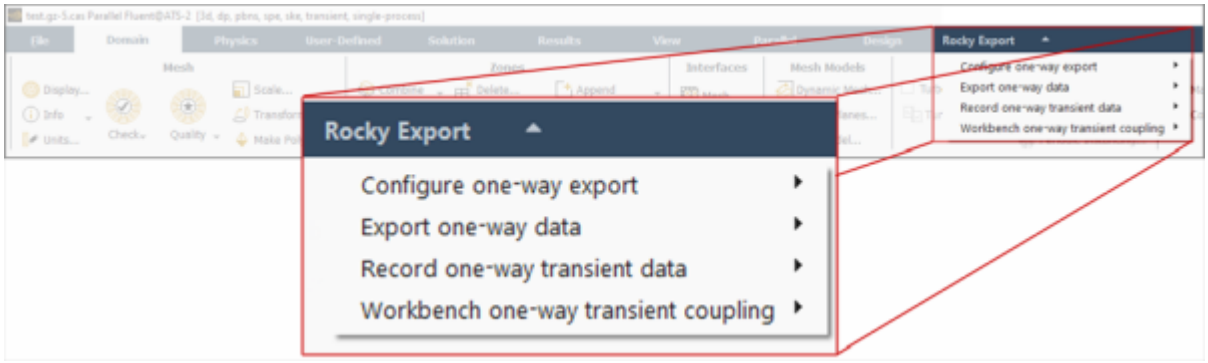


Table 1: Rocky Export menu in Fluent

Setting	Description
Configure one-way export	
Select Directory to Export	Choose the folder where the `.cas` and `.dat` files will be saved.
Select Variables to Export	Select the variables to export, such as pressure, temperature, etc.
Select Wall Thermal Zones	Define the thermal zones to export.
Set Output Frequency	Define the frequency to export transient data.
Export one-way data	
Export current data to Rocky	Export the current static field data to Rocky.

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

Setting	Description
Record one-way transient data	
Start one-way transient export	Start time recording transient one-way export data.
Stop one-way transient export	Stop time recording transient one-way export data.
Workbench one-way transient coupling	
Enable	Enable transient coupling with Workbench and Fluent (See also Set Up and Run a 1-Way Fluent Transient Project within Ansys Workbench (p. 789).).
Disable	Disable transient coupling with Workbench and Fluent (See also Set Up and Run a 1-Way Fluent Transient Project within Ansys Workbench (p. 789).).

Important:

Although 1-Way coupling does not require that Fluent be installed on the same Rocky machine running the coupled simulation, the machine exporting the fluid data out of Fluent must first have this **Ansys Fluent Coupling Support** component installed through the Rocky UI.

The **Ansys Fluent Coupling Support** component is included by default during your Rocky installation but if for some reason is not included or is in need of being updated, it can be installed later through the Rocky UI. (See also Install Ansys Coupling Components (p. 792).)

1-Way coupled simulations also require that the initial CFD simulation be set up a certain way in Fluent to ensure that it works correctly when coupled with Rocky. These required settings are outlined in the accompanying procedure Use the 1-Way Fluent Method to Process Fluent and Rocky Simulations (p. 383), and more detailed information about the Fluent settings can be found in the **CFD Coupling Technical Manual** manual (from the Rocky **Help** menu, point to **Manuals** and then click **CFD Coupling Technical Manual**.)

ABOUT THE FILES EXPORTED OUT OF FLUENT

In Fluent, there are separate steps for exporting steady-state fluid data, or recording transient fluid data. Both processes result in a special F2R file that gets saved to the same folder at the initial Fluent CAS file (or alternate folder if you choose to define one). This is the file you will import into Rocky during the 1-Way Fluent coupling process.

If the Fluent case has geometry motions, this information will also be included in the F2R file. Rocky will automatically create new Motion Frames from the F2R file in this case. (You will still need to assign those motions to the geometries later.)

In addition to the F2R file, these other files are also exported out of Fluent:

- One .dat file is saved for every time step to specify the values of the specified flow field.

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

- If the mesh position does not change between outputs, then one mesh.dat file is saved to specify the CFD mesh position. Otherwise, if the mesh position changes between outputs, additional mesh.dat files are saved for every time step.
- The geometry components used in the Fluent case file are exported to STL files. These STL files (or the CAS file itself) can be later imported into Rocky to ensure consistency between the two programs.

Important:

For 1-Way transient cases especially, the quantity of the .dat and .mesh.dat files can add up quickly. Ensure that you have enough storage space designated for the exported files, and that you are recording only the portion of the fluid data that you require for your analysis. Starting the recording after some fluid results have been processed, and recording only one full revolution of your case are a few ways you can help limit your file size.

HOW ROCKY CALCULATES FLUID-TO-PARTICLE INTERACTIONS

After importing the Fluent F2R file into Rocky, Rocky will then display (in a 3D View window) a node at all the cell centroids of the imported Fluent mesh. And will create new Motion Frames representing the geometry motions, if the original Fluent case had them.

You can then define what Laws you want applied to the imported fluid, and set a Start Time of when you want the fluid flow to be included in the coupled simulation. After the simulation is processed, Rocky will use this information to calculate how the particle flow will change because of the fluid forces.

For steady-state flows, the fluid remains constant after the Start Time is reached. For transient flows, the flow will be transient after the Start Time is reached until the end of the transient recording, and then the flow will remain constant from that point on. Tip:To achieve a steadier flow, you can offset the Start Time with negative numbers.

The influence of the fluid flow on the particle motion is achieved by adding up all the fluid forces acting on the particle, such as drag and pressure forces, as well as non-drag forces like virtual mass force, Basset force, lift force, Saffman and Magnum forces (due to particle rotation), among others. Depending on the flow conditions, most of these forces can be ignored.

A drag force exerted on a spherical (p. 1395) particle is different from a drag force that acts on a *shaped particle*. In this 1-Way Fluent coupling, various drag correlations are available based on particle type (spherical or shaped).

Because the fluid flow calculations are already complete, there is no additional processing load for Rocky when choosing this option.

ABOUT SIMULATING CONVECTIVE HEAT TRANSFER USING 1-WAY FLUENT COUPLING

When setting up the initial CFD simulation in Fluent, you can also enable heat transfer calculations, so that particles will change their temperature because of the heat transfer with the fluid. In order to compute these values, Rocky will require some thermal properties from the fluid; in order to get those properties, the Thermal Model must also be enabled in Rocky before the coupled simulation

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

is processed. (See also Enable Thermal Modeling Calculations (p. 400).) In this way, convective heat transfer between particles and fluids can be simulated.

Note:

In this version of Rocky, there are no restrictions on the type of **Energy Equation** you define in Fluent.

Tip:

To see a walk-through example of coupling with the 1-Way Fluent using steady state data, including convective heat transfer, refer to the following Workshop: Tutorial - Windshifter in the *Rocky Tutorial Guide*.

ABOUT COUPLED SIMULATIONS WITH "SHADOW BOUNDARIES"

A shadow boundary is an internal boundary without thickness. Due to the way Rocky optimizes the representation of the CFD domain in order to increase the simulation speed, in rare situations particles may be placed in a neighboring cell when the L-E mapping is performed. This is not a problem in the great majority of cases; however, when such a thing happens near a shadow boundary, there is a small chance that the particle is misplaced on the wrong side of the boundary. In order to avoid this chance occurrence, Rocky will prevent coupled simulations from processing if they have polygonal cells at shadow boundaries.

To get around this limitation, add a thickness to your shadow boundaries, or avoid using polyhedron meshes around them.

ABOUT VISUALIZING 1-WAY FLUID AND PARTICLE FLOWS

You can choose to visualize in a 3D View window the fluid flow data represented by Nodes and/or Vectors. This display, when combined with Particle visualization, can provide you a fuller visual confirmation of your 1-Way coupled simulation (Figure 2).

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

Figure 3.193: Example of a (steady state) 1-Way Fluent simulation showing static fluid flow vectors (in blue), and transient particle nodes + vectors (in red)



(See also About Using the Coloring Tab to Change a 3D View (p. 1074) and About Properties (p. 933).)

OTHER LIMITATIONS

Shell (p. 1395) and Solid (p. 1395) particles composed of Multiple Elements (also known as meshed (p. 1392) or flexible particles) are incompatible with 1-Way Fluent coupling. The only flexible particle shapes compatible with this feature are Fibers (p. 1391). (See also About Adding and Editing Particle Sets (p. 602).)

In addition, Shell (p. 1395) particles composed of Single Elements (also known as "rigid" particles) are also incompatible with 1-Way Fluent coupling.

Also, note that when combined with Coarse Grain models, only drag forces are considered. This means that other non-drag fluid force models, such as virtual mass and lift, are incompatible with Coarse Grain modeling. Tip:To see a walk-through example of 1-Way Fluent in use with Coarse Grain Modeling, refer to Tutorial - Mixing Tee in the *Rocky Tutorial Guide*.

For a full list of limitations, refer to the Ansys and CFD Coupling Limitations (p. 63) topic.

ROCKY EXPORT MENU OPTIONS (IN FLUENT)

Refer to Figure 1 above, as well as Figures 3-4 and the table below to help you understand the Rocky Export options in Fluent.

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

Figure 3.194: The Additional Variables Selector dialog in Fluent (left). These become custom Properties in Rocky after F2R import (right).

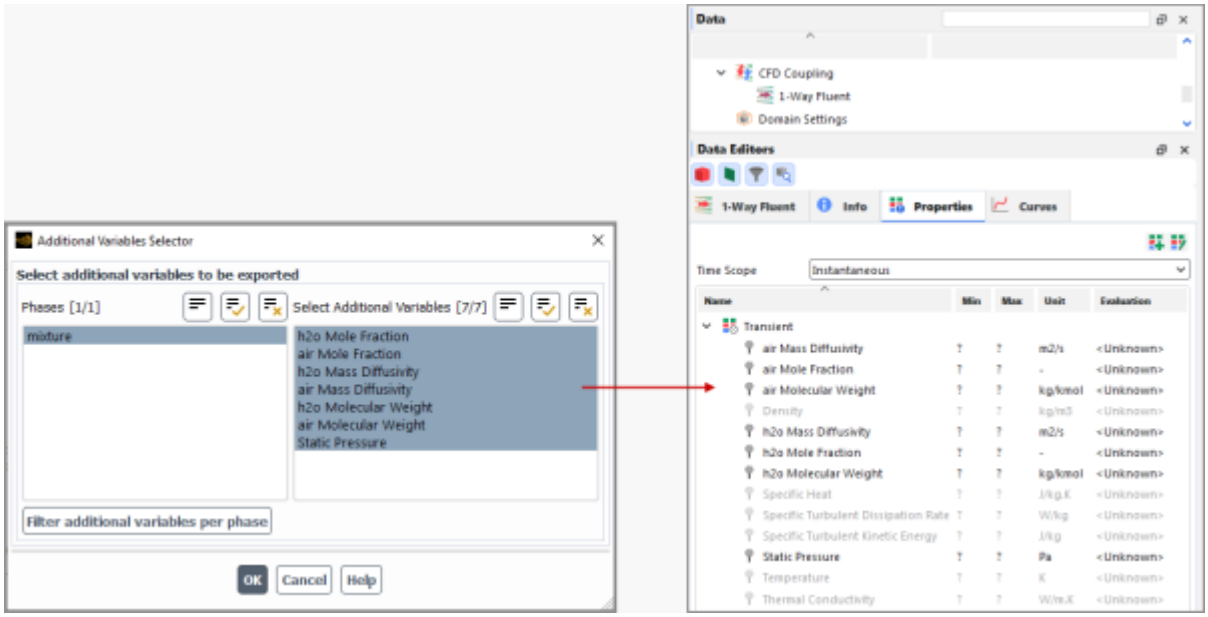


Figure 3.195: The Fluent to Rocky (F2R) time-step multiplier dialog in Fluent

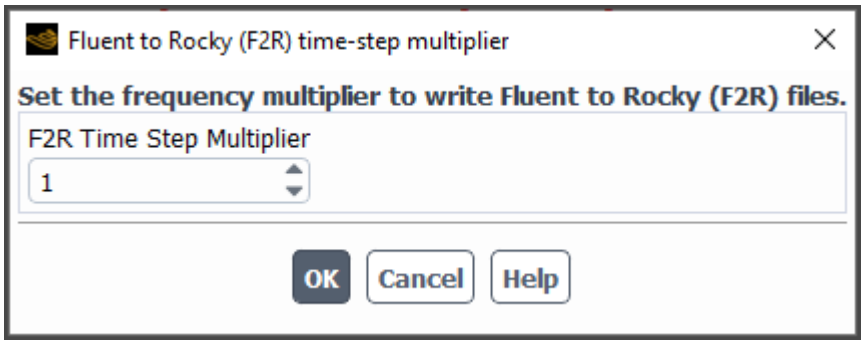


Table 1: Rocky Export menu options (in Fluent)

Menu Option	Description
Configure one-way export	
Select Directory to Export	Enables you to select where the exported F2R and related files will be saved. Note: If this is not defined, Fluent will automatically use the same folder as your Fluent case file. Important: Especially for transient files, the size and quantity of the exported files can add up quickly. Ensure that the directory you select has sufficient storage space.
Select Variables to Export	Opens the **Additional Variables Selector** dialog (Figure 3, above), which enables you to define what Fluent variables—such as species, for example—you want included in the exported files. Each variable will come across as a separate Property that you can use in Rocky for post-processing (Figure 3, above). Because the variables can increase the size of the DAT files, limiting what variables you export to only what you require can help you control file size.

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

Menu Option	Description
Set Output Frequency	Opens the **Fluent to Rocky (F2R) time-step multiplier** dialog (Figure 4 above), which enables you to define how often you want the export process to save fluid files. For example, a **F2R Time Step Multiplier** of: • 1 means that one set of fluid files will be saved for every Fluent Time Step. • 2 means that one set of fluid files will be saved for every 2 Fluent Time Steps.
Export one-way data	
Export current data to Rocky	For steady-state flows, exports fluid data at the currently selected Fluent Time Step. The resulting F2R file can then be imported into Rocky as part of the 1-Way Fluent coupling method.
Record one-way transient data	
Start one-way transient export	For transient flows, will begin the recording of fluid data at that point in the Fluent case processing. Tips: • It is recommended that you process your Fluent case until the point you want fluid data recorded, stop processing your case, select this option to start recording, and then resume processing your Fluent case. • To start the recording from a condition without results, ensure you first initialize the solution before you select this option.
Stop one-way transient export	For transient flows, will stop the recording of fluid data at that point in the Fluent case. The resulting F2R file can then be imported into Rocky as part of the 1-Way Fluent coupling method. Tip:It is recommended that you stop processing your Fluent case after the fluid data you want is calculated, and then select this option to stop recording your transient data.
Workbench one-way transient coupling	
Enable	For transient flows, enables you to set up and process the Rocky-Fluent coupled case through Ansys Workbench. (See also Set Up and Run a 1-Way Fluent Transient Project within Ansys Workbench (p. 789)).
Disable	For transient flows, disables the ability to set up and process the Rocky-Fluent coupled case through Ansys Workbench.

1-WAY FLUENT PARAMETERS (IN ROCKY)

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

See the images and table below to help you understand the various properties you can set for the 1-Way Fluent method. For more information on this method, you may also refer to the CFD Coupling Technical Manual document included within the program. (From the **Help** menu, point to **Manuals**, and then click **CFD Coupling Technical Manual**.)

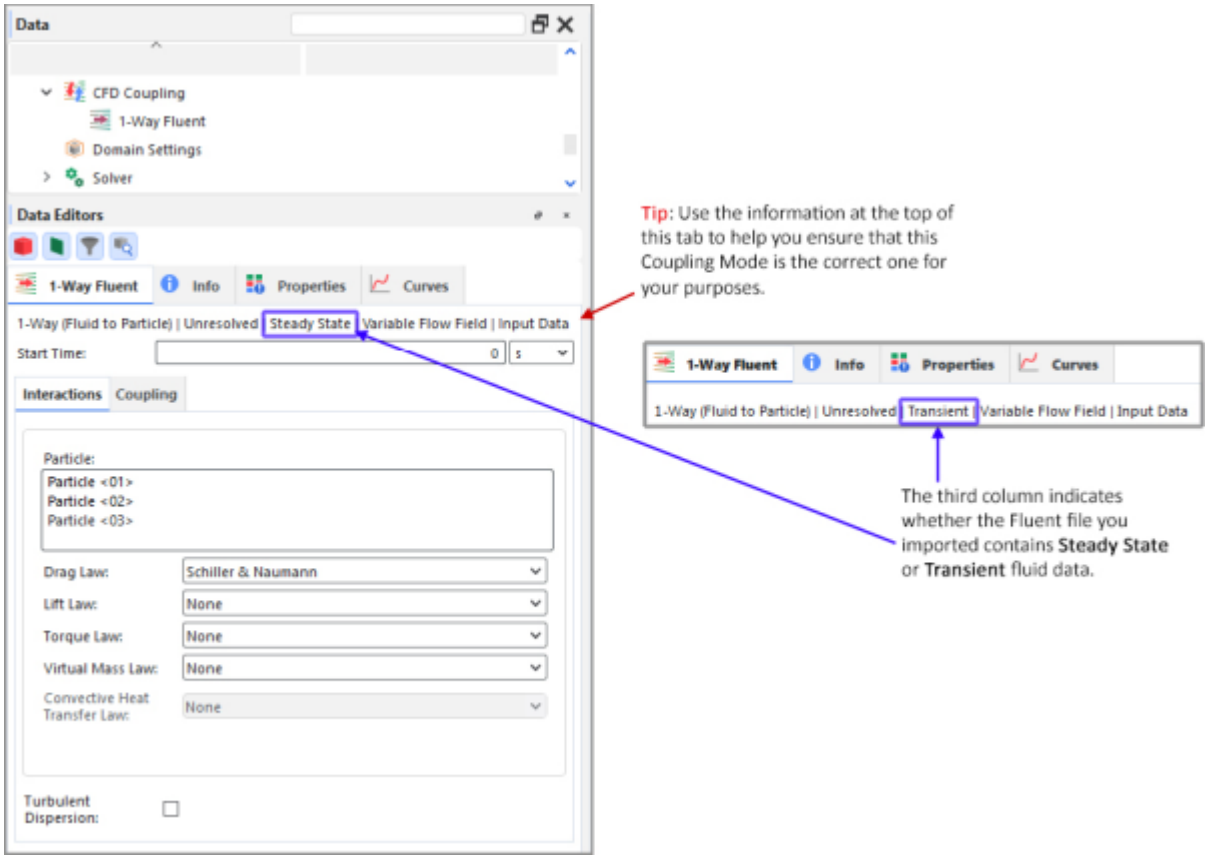


Figure 5 : CFD Coupling, 1-Way Fluent in the Data Editors panel

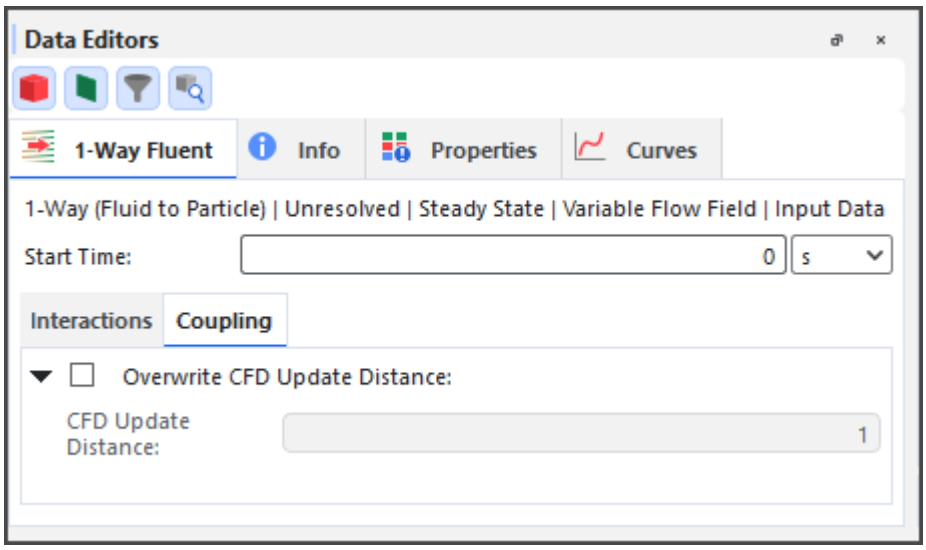


Figure 6 : CFD Coupling, 1-Way Fluent, Coupling Tab in the Data Editors panel when Advanced Features are enabled

Table 2: 1-Way Fluent Settings (in Rocky, all tabs)

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

Setting	Description	Range
Start Time	The time during the simulation when you want to start taking fluid flow into account.	Any value
Interactions tab		
Particle	Permits correlations- such as drag, lift, torque, virtual mass, and convective heat transfer laws (if applicable)- to be defined per *Particle set* (p. 1393). You accomplish this by selecting (or multi-selecting) from a list of previously defined Particle sets, and then defining the applicable CFD Laws for only the selected set (or sets). **Important:** Even though they will still be listed here, neither flexible *Shell* (p. 1395) nor flexible *Solid* (p. 1395) particle shapes are supported in CFD Coupling approaches, including this one. (See also Ansys and CFD Coupling Limitations (p. 63)).	Automatically provided
Drag Law	Correlation used by Rocky to calculate drag forces on particles (with the exception of *Fibers* (p. 1391) and also *Solid* (p. 1395) Sphero-Cylinder shapes-see note below): • **Schiller & Naumann**: Recommended for Spheres only. • **Haider & Levenspiel**: Recommended for Spheres and isometric *shaped particle* (p. 1395) since it takes the particle sphericity into account in order to calculate the drag coefficient. • **Ganser**: Recommended for both spherical and shaped particles since it considers the effect of shape and alignment of the particle with the flow when computing the drag coefficient. Note: Ganser is incompatible with *Assembly* (p. 1389), *concave* (p. 1390), and *Shell* (p. 1395) particle shapes. (See also About Adding and Editing Particle Sets (p. 602)). • **Dallavalle**: Recommended for Spheres only. Although it yields results similar to the **Schiller & Naumann** drag law, as a continuous function, it can lead to more stable simulations on very sensitive cases.	Schiller & Naumann; Haider & Levenspiel; Ganser; Dallavalle: Marheineke & Wegener; Morsi & Alexander Note: If you have an *external Module* that adds in an additional CFD law, you must use that law at least once in your simulation project if the Module is enabled. Refer to the Module's documentation (if provided) for more information. (See also Rocky Simulation Entities that can be Affected by Modules (p. 223)).

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

Setting	Description	Range
	• **Marheineke & Wegener**: The *only* drag law currently recommended for use with *Fibers* (p. 1391) and also *Solid* (p. 1395) Sphero-Cylinder particle shapes. • **Morsi & Alexander**: Recommended for Spheres, but can be adapted to shaped particles when the adjustable coefficients are used. Best used in cases where there is a desire to tune the drag law to match experimental results. **Note:** Unless stated otherwise, the above drag laws apply to any supported particle shapes (see also Ansys and CFD Coupling Limitations (p. 63)) except *Fibers* (p. 1391) and also *Solid* (p. 1395) Sphero-Cylinder shapes, which have their own modified drag law based on the work by Marheineke and Wegener that should always be used for those shapes. **Tip:** For more information about these laws, refer to the CFD Coupling Technical Manual. (From the Rocky **Help** menu, point to **Manuals**, and then click **CFD Coupling Technical Manual**.)	
Lift Law	Correlation used by Rocky to calculate the fluid-particle force transversal to the direction of the relative velocity.	None; Saffman; Mei Note: If you have an *external Module* that adds in an additional CFD law, you must use that law at least once in your simulation project if the Module is enabled. Refer to the Module's documentation (if provided) for more information. (See also Rocky Simulation Entities that can be Affected by Modules (p. 223)).

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

Setting	Description	Range
Torque Law	Correlation used by Rocky to calculate the torque exerted by the fluid over the particle.	None; Dennis, Singh & Ingham Note: If you have an *external Module* that adds in an additional CFD law, you must use that law at least once in your simulation project if the Module is enabled. Refer to the Module's documentation (if provided) for more information. (See also Rocky Simulation Entities that can be Affected by Modules (p. 223)).
Virtual Mass Law	Correlation used by Rocky to calculate the virtual mass coefficient.	None; Constant Note: If you have an *external Module* that adds in an additional CFD law, you must use that law at least once in your simulation project if the Module is enabled. Refer to the Module's documentation (if provided) for more information. (See also Rocky Simulation Entities that can be Affected by Modules (p. 223)).
Convective Heat Transfer Law	When both **Thermal Model** is enabled (see also About Physics Parameters (p. 417)) and the Fluent exported results have thermal solution data, this correlation is used by Rocky to calculate the fluid-particle convective heat transfer coefficient.	None; Ranz & Marshall; Whitaker Note: If you have an *external Module* (p. 1391) that adds in an additional CFD law, you must use that law at least once in your simulation project if the Module is enabled. Refer to the Module's documentation (if provided) for more information. (See also Rocky Simulation Entities that can be Affected by Modules (p. 223)).

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

Setting	Description	Range
Turbulent Dispersion	When the imported .f2r file has a turbulent model that uses turbulence kinetic energy as one of its quantities, this allows those turbulent parameters to be used and modified in Rocky. Clearing this checkbox prevents the turbulent parameters from being used during the Rocky simulation. **Tip:** For more information about this parameter, refer to the CFD Coupling Technical Manual. (From the Rocky **Help** menu, point to **Manuals**, and then click **CFD Coupling Technical Manual**.)	Turns on or off
Morsi & Alexander Parameters Use Defined Constants	When **Morsi & Alexander** is selected for **Drag Law**, this enables you to determine how the model's three coefficients will be determined. Specifically: • When cleared, the standard values provided with the model will be used. See the *CFD Coupling Technical Manual* for more detail. (From the Rocky **Help** menu, point to **Manuals**, and then click *CFD Coupling Technical Manual*.) • When enabled, you can define your own set of coefficients that will be used by the model. Tip: Defining your own constants can be particularly useful in cases where you have experimental data measuring the drag of fluid over particles that you want matched in the simulation by adjusting the coefficients of the drag law.	Turns on or off
K_1	When **Use Defined Constants** is enabled, this is the value you want assigned to the K_1 constant used in the **Morsi & Alexander** drag law calculation. See also the *CFD Coupling Technical Manual*. (From the Rocky **Help** menu, point to **Manuals**, and then click *CFD Coupling Technical Manual*.)	Any value
K_2	When **Use Defined Constants** is enabled, this is the value you want assigned to the K_2 constant used in the **Morsi & Alexander** drag law calculation. See also the *CFD Coupling Technical Manual*. (From the Rocky **Help** menu, point to **Manuals**, and then click *CFD Coupling Technical Manual*.)	Any value
K_3	When **Use Defined Constants** is enabled, this is the value you want assigned to the K_3 constant used in the **Morsi & Alexander** drag law calculation. See	Any value

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

Setting	Description	Range
	also the *CFD Coupling Technical Manual*. (From the Rocky **Help** menu, point to **Manuals**, and then click *CFD Coupling Technical Manual*.)	
Coupling tab		
Overwrite CFD Update Distance	When the **Advanced Features** checkbox is enabled on the **Options | Preferences** dialog (see also About Setting Global Preferences (p. 227)), this allows you to overwrite the default distance used as a criteria for updating the volume fraction field. As this step may take a considerable amount of time, increasing the associated **CFD Update Distance** value (and therefore reducing the frequency at which the volume fraction is updated) might help reduce processing time.	Turns on or off
CFD Update Distance	When the **Advanced Features** checkbox is enabled on the **Options | Preferences** dialog (see also About Setting Global Preferences), and the **Overwrite CFD Update Distance** checkbox is enabled, this defines the maximum distance the particles are allowed to travel before a volume fraction field update. The smaller the value, the more frequent the update. As this step may take a considerable amount of time, increasing the update distance (and therefore reducing the frequency at which the volume fraction is updated) might help reduce processing time.	Positive values

What would you like to do?

- Use the 1-Way Fluent Method to Process Fluent and Rocky Simulations (p. 383)
- Import Fluid Flow Data from Ansys Fluent (p. 788)
- Set Up and Run a 1-Way Fluent Transient Project within Ansys Workbench (p. 789)

See Also:

- Set or Modify Fluid and/or Air Flow Properties (p. 757)
- Set Simulation Parameters (p. 414)
- I cannot find Particle Size Scale Factor (p. 1330)

Import Fluid Flow Data from Ansys Fluent

Important:

This procedure covers only the 1-Way Fluent coupling steps that are required in the Rocky UI. For a full coupling procedure, including steps that need to be taken in Ansys Fluent,

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

see the Use the 1-Way Fluent Method to Process Fluent and Rocky Simulations (p. 383) procedure.

Note:

If you are not certain whether you have installed the **Ansys Fluent Coupling Support** component during your Rocky installation, first follow the Install Ansys Coupling Components (p. 792) procedure before beginning these steps.

1. From the **Data** panel, click **CFD Coupling** and then from the **Data Editors** panel, choose **Fluent (Fluid → Particle)** from the **Coupling Mode** list.
2. From the **Select Fluent 2 Rocky export file** dialog, locate and select the F2R file you want to use, and then click **Open**. Important: For F2R files that include transient data, this step could take several minutes to complete. In addition, if you chose to set up Moving Meshes in Fluent, one new Frame will appear under **Motion Frames** in the **Data** panel for each Cell Zone with motions you have defined. Important: If you have defined your Moving Meshes in Fluent using expressions, Rocky will not be able to create the associated Motion Frames for you automatically. Rather, you must manually create your own Motion Frames in Rocky to match the Fluent motions as closely as possible. Tip:To save loading time, the Nodes for the imported fluid are turned off by default in the 3D View. To see them, from the Coloring tab, enable the **Nodes** checkbox. (See also About Using the Coloring Tab to Change a 3D View.)
3. Under **CFD Coupling**, select the new item and then from the **Data Editors** panel, on the **1-Way Fluent** tab, do the following:
   a. Define the **Start Time** you want.
   b. From the **Interactions** sub-tab, do the following:
      i. From the **Particle** list, select (or multi-select) the Particle set name(s) for which you want to define correlations, and then select the various CFD laws you want for the selected set(s).
      ii. Define the **Turbulent Dispersion** options you want.

   The fluid flow data will be shown interacting with particles when you process the simulation. (See also Processing a Simulation (p. 907).)

   The data is now available to visualize in a 3D View window. (See also Create and Modify a 3D View (p. 1040).)

See Also:

- About Using the 1-Way Fluent Method (p. 775)
- The CFD Coupling Technical Manual document (from the Rocky **Help** menu, point to **Manuals**, and then click **CFD Coupling Technical Manual**)

Set Up and Run a 1-Way Fluent Transient Project within Ansys Workbench

Rocky supports setting up and processing a 1-Way Fluent transient simulation with Ansys Workbench. However, there are specific steps that need to be taken to ensure this workflow.

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

Use the procedure below.

TO SET UP AND RUN A 1-WAY FLUENT TRANSIENT PROJECT USING ANSYS WORKBENCH

1. Set up the initial Workbench project by doing the following:

   a. Open Workbench and create a new project.

   b. Create a **Fluent System**.

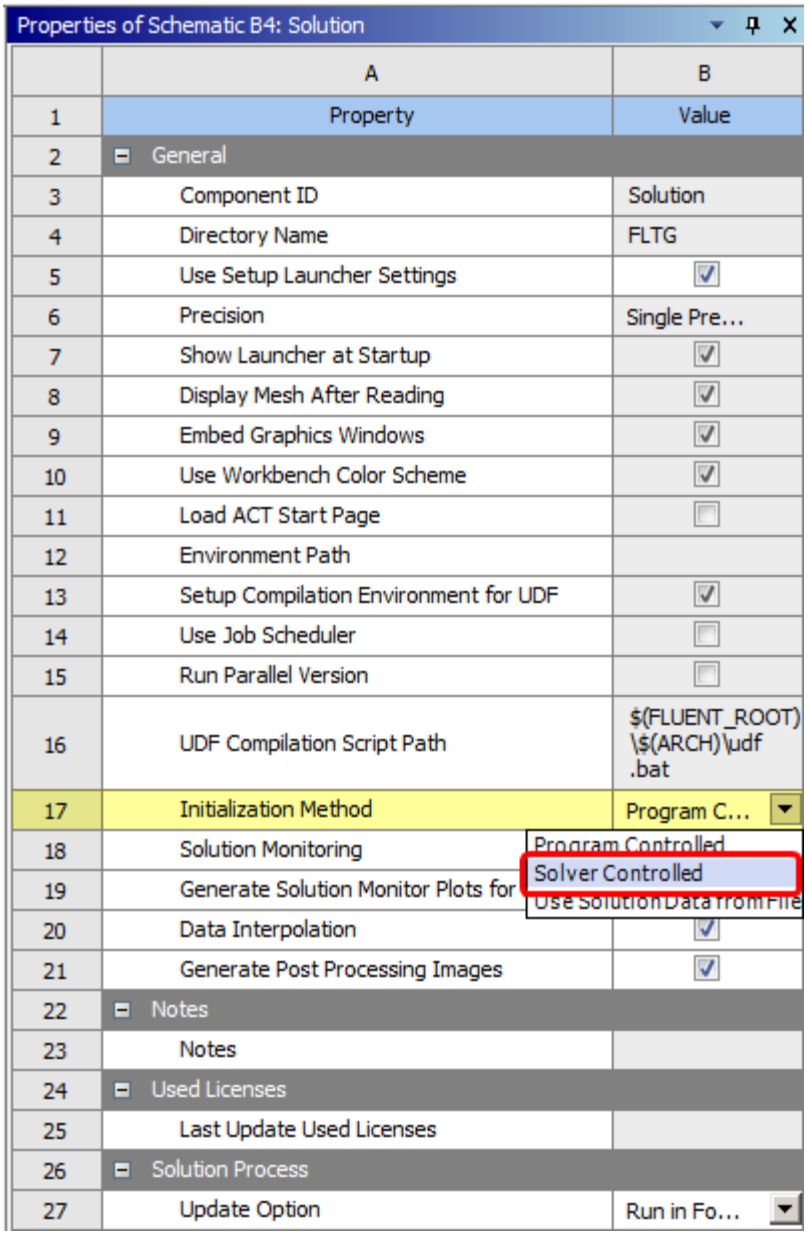
   c. Set the **Initialization Method** for that system by doing the following:

      i. From the Fluent block, right-click **Solution**, and then click **Properties**.

      ii. From the **Properties** panel, define the value for the **Initialization Method** property as **Solver Controlled** (Figure 1).

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

Figure 3.196: Properties panel for Fluent Solution



2. Set up the Fluent project by doing the following:

   a. From your Workbench project, open Fluent.

   b. Open or set up the transient project you want to couple with Rocky. (See also Use the 1-Way Fluent Method to Process Fluent and Rocky Simulations (p. 383).)

   c. Enable Workbench transient coupling by doing the following:

      i. From the **Rocky Export** menu, point to **Workbench one-way transient coupling** and then click **Enable**.

      ii. Save your project, close Fluent, and return to your Workbench project.

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

iii. Update **Fluent System** Solution in Workbench so that there are results to set up one-way transient coupling in Rocky.

3. Set up the Rocky project by doing the following:

   a. From your Workbench project, create a **Rocky** system.

   b. Connect the **Fluent System** Solution to **Rocky** System Setup in Workbench.

   c. From your Workbench project, open Rocky.

   d. Open or set up the project that you want to couple with the Fluent transient results. (See also Use the 1-Way Fluent Method to Process Fluent and Rocky Simulations (p. 383).)

   e. Save your Rocky project, close Rocky, and return to Workbench.

   f. Update the whole project in Workbench.

The next time you have modifications to your Fluent project, the 1-Way transient results can be updated from Workbench and the Fluent GUI can be closed.

See Also:

- About Using the 1-Way Fluent Method (p. 775)
- Use the 1-Way Fluent Method to Process Fluent and Rocky Simulations (p. 383)
- Use a Journal to Run Fluent and Export Transient 1-Way Results (p. 1313)

INSTALL ANSYS COUPLING COMPONENTS

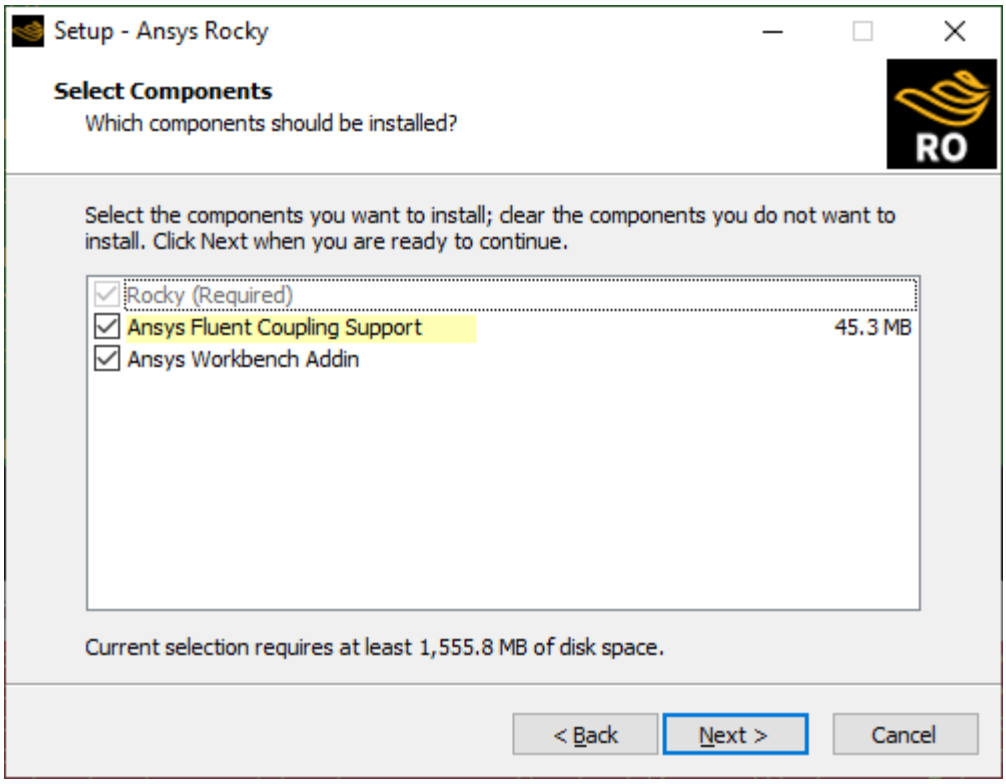
When coupling Rocky with Ansys Fluent or when using Rocky with Ansys Workbench, special components must be installed. Use the sections below to help you determine what components you need for which Ansys product, and how to install them.

INSTALL ANSYS COMPONENT FOR FLUENT COUPLING (NO WORKBENCH)

Use this procedure if you want to use the 1-Way Fluent (see also About Using the 1-Way Fluent Method (p. 775)), the 2-Way Fluent (see also About Using the 2-Way Fluent Method (p. 795)), or the 2-Way Fluent Semi-Resolved (see also About Using the 2-Way Fluent Semi-Resolved Method (p. 831)) coupling method for your simulations without going through Ansys Workbench, but are not certain whether you have installed the **Ansys Fluent Coupling Support** component during your initial Rocky installation (Figure 1).

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

Figure 3.197: The Ansys Fluent Coupling Support option selected during Rocky installation



To install Ansys coupling components for Fluent Coupling (no Workbench):

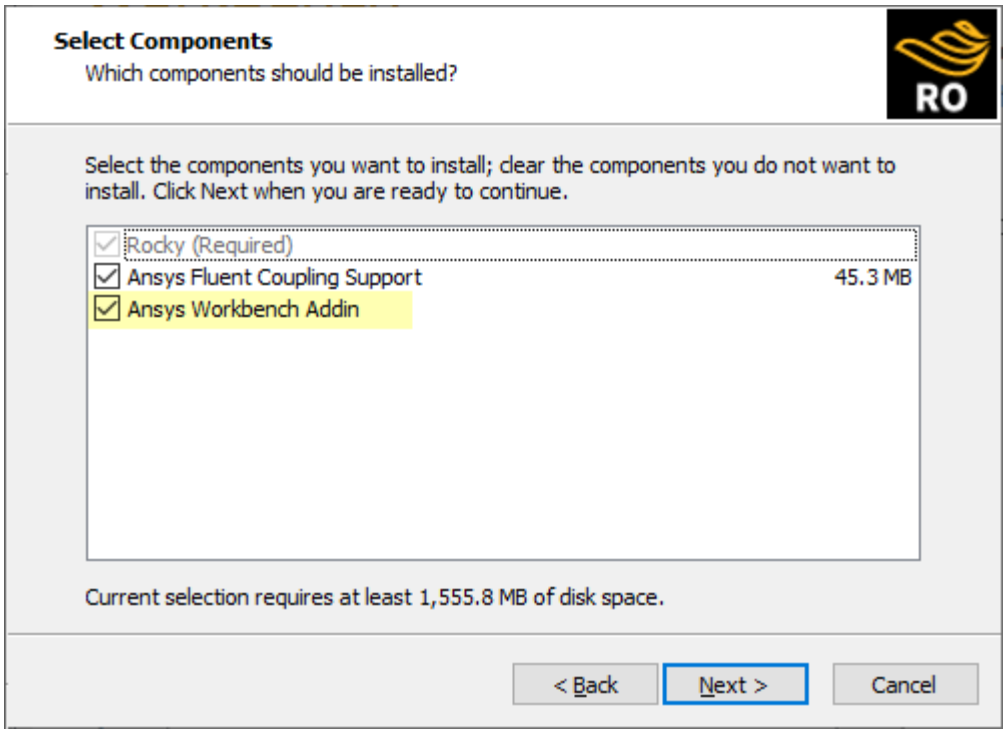
1. From the Rocky **Options** menu, point to **Ansys** and then click **Install Fluent/Rocky export**.

Install Ansys Components for 1-Way Fluent (and SpaceClaim) Coupling (through Workbench)

Use this procedure if you want to use the 1-Way Fluent (see also About Using the 1-Way Fluent Method (p. 775)) coupling method through Ansys Workbench (which also requires a geometry from Ansys SpaceClaim) but are not certain whether you have installed both the **Ansys Fluent Coupling Support** (Figure 1) and **Ansys Workbench Addin** (Figure 2) components during your initial Rocky installation.

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

Figure 3.198: Rocky install screen showing Ansys Workbench Addin selected



To install Ansys coupling components for 1-Way Fluent (and SpaceClaim) Coupling (Workbench):

1. For Windows machines, reinstall Rocky and ensure that both the **Ansys Fluent Coupling Support** and the **Ansys Workbench Addin** components remain selected during installation.

INSTALL ANSYS COMPONENTS FOR MECHANICAL (AND ANSYS SPACECLAIM) COUPLING (THROUGH WORKBENCH)

Use this procedure if you want to use Ansys Mechanical through Workbench (which also requires a geometry from Ansys SpaceClaim, see also About Rocky and Ansys Mechanical Integration (p. 170)) with Rocky, and are unsure whether you have installed **Ansys Workbench Addin** (Figure 2) component during your initial Rocky installation.

To install Ansys coupling components for Mechanical (and Ansys SpaceClaim) 1-Way Coupling (Workbench):

1. For Windows machines, reinstall Rocky and ensure that the **Ansys Workbench Addin** component remains selected during installation.

See Also:

- About Using the 1-Way Fluent Method (p. 775)
- Set Up and Run a 1-Way Fluent Transient Project within Ansys Workbench (p. 789)
- About Using the 2-Way Fluent Method (p. 795)
- About Using the 2-Way Fluent Semi-Resolved Method (p. 831)
- About Rocky and Ansys Mechanical Integration (p. 170)

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

- I cannot find the Rocky component in Workbench (p. 1327)

Uninstall Ansys Coupling Components

1. From the Rocky **Options** menu, point to **Ansys** and then click **Uninstall Fluent/Rocky export**.

See Also:

- About Using the 1-Way Fluent Method (p. 775)
- Install Ansys Coupling Components (p. 792)
- Remove Rocky (p. 1296)

USE THE 2-WAY FLUENT METHOD

The 2-Way Fluent method enables you to both send Rocky particle flow data to and accept fluid data from Ansys Fluent. It is considered to be an unresolved method.

Note:

There are no special components that must be installed in Rocky for conducting a 2-Way coupled simulation with Fluent.

What would you like to do?

- Learn more About Using the 2-Way Fluent Method (p. 795)
- Exchange Rocky Particle Data with Fluid Flow Data from Ansys Fluent (p. 827), which covers only the final coupling portion of the entire 2-Way Fluent Coupling procedure.
- Use the 2-Way Fluent Method to Process a Fluent and Rocky Simulation (p. 386), which covers the full procedure, including the setup steps that are required in Fluent.

See Also:

- Set or Modify Fluid and/or Air Flow Properties (p. 757)
- Use the 1-Way LBM Method (p. 770)
- Use the 1-Way Constant Method (p. 763)
- Use the 1-Way Fluent Method (p. 775)
- Use the 2-Way Fluent Semi-Resolved Method (p. 830)
- Set Simulation Parameters (p. 414)

About Using the 2-Way Fluent Method

The 2-Way Fluent Method is an unresolved CFD Coupling option that enables Rocky to receive fluid flow data from Ansys Fluent, calculate how the fluid movement affects the particle flow, and then send the momentum (and other quantities) sources back to Fluent for Fluent to calculate the fluid

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

flow for the next time step. 2-Way Fluent coupling results in a Rocky simulation that fully depicts how particles and fluids interact with each other in the confines of the equipment.

Tip:

If your application requires a very refined Fluent mesh in order to accurately simulate the fluid flow behavior, and the resulting fluid cell sizes are many times smaller than the particle size itself, you might want to instead consider using the 2-Way Fluent Semi-Resolved Method (p. 831).

SPECIAL REQUIREMENTS FOR 2-WAY FLUENT COUPLING

To make it possible to share the Fluent fluid flow data with Rocky, Rocky includes a special **Ansys Fluent Coupling Support** component. This component is included by default during your Rocky installation but if for some reason is not included or is in need of being updated, it can be installed later through the Rocky UI. (See also Install Ansys Coupling Components (p. 792).)

In addition, 2-Way coupling requires that a Rocky-supported version of Ansys Fluent is installed upon the same machine that contains your Rocky program installation. (See also System Requirements (p. 67).)

FLUENT CASE REQUIREMENTS AND SUPPORTED FEATURES

2-Way Fluent coupled simulations require that the initial CFD simulation be set up a certain way in Fluent to ensure that it works correctly when coupled with Rocky. These required settings are outlined in the accompanying procedure Use the 2-Way Fluent Method to Process Fluent and Rocky Simulations (p. 386), and more detailed information about the Fluent settings can be found in the **CFD Coupling Technical Manual** manual (from the Rocky **Help** menu, point to **Manuals** and then click **CFD Coupling Technical Manual**.)

There are many ways you can customize your coupled simulations. For example, you can measure heat exchange by using thermal modeling, can choose to begin your simulation empty or with particles and/or fluid velocities already in the system, and can use the same or different geometry files in both programs.

Note:

In this version of Rocky, there are no restrictions on the type of **Energy Equation** you define in Fluent.

Use the sections below to learn more about setting up your case in Fluent for later 2-way coupling with Rocky.

SUPPORT FOR FLUID PHASES

Both single phase and multiphase approaches are supported using the 2-Way Fluent CFD Coupling method. Single phase (p. 1395) is defined as one CFD fluid phase plus one DEM particle phase. Multiphase (p. 1392) (or multi-phase) is defined as two or more CFD fluid phases plus one DEM particle phase.

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

Important: Since these phases are defined in Ansys Fluent, and this user manual's purpose is to cover only Rocky product usage, most of the information provided in this document will apply to *both* approaches. However, differences will be specified when critical to the understanding of Rocky-Fluent product interactions.

About the Single Phase Approach The single phase approach is faster than the equivalent simulation using the multiphase approach. It is also easier to set up and allows for a broader range of models.

About the Multiphase Approach In this version of Rocky, the following three types of **Turbulence Multiphase Model** you define for your multiphase Fluent case are supported in 2-Way Fluent simulations:

- Mixture
- Dispersed
- Per Phase

These turbulence models are also compatible with sub-stepping. (See **Sub-Stepping** definition in Table 2 below.)

SUPPORT FOR UDMS AND UDFS

User Defined Memories (UDMs) are supported, as well as mesh interface zones. If a Fluent case has additional UDFs beyond those used for Rocky, Rocky will copy (and if only the source files are provided, compile) the additional UDFs to the coupled project as long as the below requirements are met. (See also additional information in the Rocky **CFD Coupling Technical Manual**.)

UDF Usage Requirements: There are several requirements in order to make use of UDFs, as listed below:

1. You must place all UDF files in the same folder in which your Fluent CAS (CAS.H5 or CAS.GZ) file is located. Rocky will then copy those files to the simulation folder and use them during the 2-Way coupling simulation.
2. You must provide the UDF files via one of the following two methods:
    a. By providing the source files. (E.g., the .c, .h, and .cpp source and header files.) In this case, you must launch Rocky from an environment that has all the necessary variables for UDF compilation.

> **Tip:**
>
> An easy way to accomplish this is to open a command prompt and then call the `udf.bat` file, which in Windows, is usually located in the following folder `C:Program Files\ANSYS Inc\ {version}\fluent\ntbin\win64` Following this step, you can then launch Rocky from the command prompt. (See also Appendix G: Perform Tasks from the Command Line or Journal (p. 1308).)

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

b. **Recommended:** By providing the folder containing the UDF binary files, you must have already compiled the UDF and there are no requirements in terms of the environment variables.

Important:

Ensure that you do not change nor replace any rocky source functions, as this might affect the coupling behavior.

ABOUT SUPPORTING SOURCE TERMS DEFINED IN FLUENT

In this version of Rocky, you may define within your Fluent CAS (CAS.H5 or CAS.GZ) file source terms, such as momentum, via journal. During the coupled simulation, Rocky will keep these source terms and will add its own source terms on top of the ones you defined in Fluent.

ABOUT SUPPORTING FLUENT MOVING MESHES

Rocky will also support importing multiple Fluent Moving Meshes during the setup of a 2-Way Fluent coupling simulation, which enables you to see simple translation or rotation movements on both the Rocky and Fluent sides. (See also About Creating and Applying Motion Frames (p. 533).)

Note:

In this version of Rocky, moving meshes are now compatible with sub-stepping. (See also **Sub-Stepping** definition in Table 2 below.)

SUPPORT FOR MULTIPLE FLUID SPECIES AND MASS SOURCE TERMS

This version of Rocky supports 2-Way coupling with Fluent cases that are defined with multiple species (multi-species). In these cases, Fluent shares species information with Rocky and Rocky sends the resulting mass source terms back to Fluent. This allows you to simulate particle-fluid interactions on the individual chemical components within a fluid mixture, thereby making it possible for you to study chemical reactions such as combustion, crystallization, and more.

This ability is possible only when used in combination with an external module (p. 1391) that supports these computations. This can be a ready-to-use module—such as the *CFD-Coupled Particle Crystallization* or the *CFD-Coupled Particle Combustion* module—or a module you create yourself using Rocky's Solver SDK functionality. (To learn more, from the **Help** menu, point to **Manuals** and then click **Solver SDK Manual**.)

As illustrated in Figure 1, when multiple species are enabled within your Fluent case, the following three variables are sent from Fluent to Rocky per species:

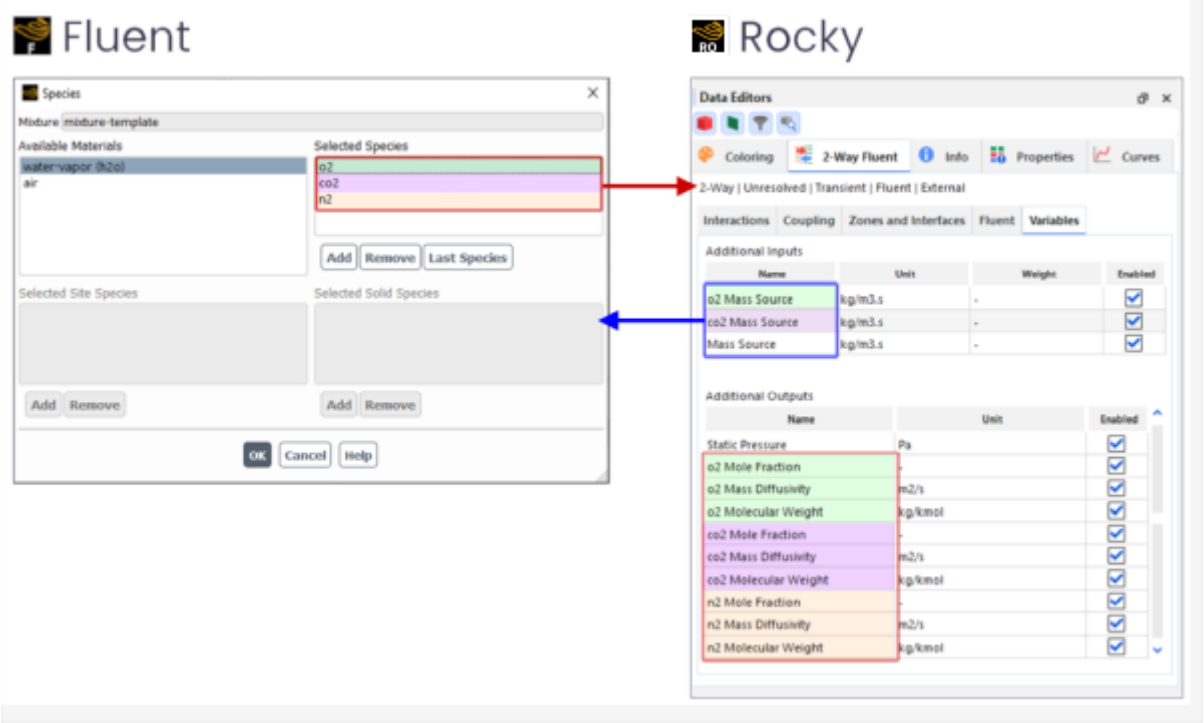
- Mole Fraction
- Mass Diffusivity
- Molecular Weight

During processing, Rocky sends back to Fluent mass source terms for all but the *last species* that you specified in the Fluent case, and Fluent will use this information to close the mass balance. Thus, what you set for the *last species* in your Fluent setup is very important as Fluent considers it to be

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

the bulk species. This *last species*, therefore, will not result in a mass source term from Rocky. If needed, you can infer the mass source for the final species by subtracting the others from the total.

Figure 3.199: Illustration showing Fluent species information (left, red box) being shared with Rocky (right, red box) during 2-Way Fluent setup. Later during processing, Rocky sends mass source terms (right, blue box) back to Fluent for all but the last species in the list.



For more information about these variables, see the *About Additional Input and Output Variables* section below.

Tip:

For a brief overview of this functionality, refer to the video *Simulate Chemical Reactions in Rocky* on the Tips and Tricks page of the Customer Portal.

ABOUT THE FILES AND FORMATS USED IN 2-WAY FLUENT COUPLING

When setting 2-Way Fluent CFD Coupling, Rocky will ask you for two kinds of files that you create as part of your initial Fluent setup: a CAS (CAS.H5 or CAS.GZ) file, which contains the initial CFD simulation setting you created in Fluent, and an optional DAT (DAT.H5 or DAT.GZ) file, which contains initial flow field data for the fluid.

Since Fluent 2020 R1, the standard format (Common Fluid Format (CFF)) for case files is CAS . H5 and DAT . H5. However, these files are not as fully compatible with post-processing in Fluent as were the legacy format of CAS . GZ and DAT . GZ. Therefore, an additional file with the format FLPRJ is required by CFD-Post in order to load the transient results from the Fluent simulation. This is achieved by using Fluent's auto-save feature, which writes the .cas.h5, .dat.h5, and also the .flprj files, all of which ensures the files are fully compatible with CFD-Post.

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

During the coupled simulation, Rocky will write another set of CAS.H5 and DAT.H5 files that allow you to restart and resume the Rocky simulation. Specifically, when you provide Rocky with the Fluent files, Rocky makes copies of them in an internal "file" folder within your project folder, and renames them: the CAS (CAS.H5 or CAS.GZ) file becomes "two_way_coupling" and the DAT (DAT.H5 or DAT.GZ) file becomes "with_particles". Therefore, the filenames `two_way_coupling.cas`, `two_way_coupling.cas.h5`, and `two_way_coupling.cas.gz` are reserved names by Rocky and they are not allowed in the Fluent auto-save function. Important: Be aware that Fluent case files with the aforementioned reserved names will be blocked. Tip:In order to save space in your storage system and not duplicate the CAS.H5 and DAT.H5 files unnecessarily, it is recommended that you disable the **Keep all files** option and keep only the last 2 (or more) files. (See also the **Fluent Files and Storage** section below.)

Later when you run or process the coupled simulation, Rocky adds components to these files and moves them to another "rocky\files" folder within your project folder. These are the files that will be used to generate the coupled simulation.

If at some point you want to modify the CAS (CAS.H5 or CAS.GZ) file, you can do so one of two ways:

- From inside Rocky, you can click **Open** on the **2-Way Fluent** | **Fluent** tab, which opens the Rocky "two_way_coupling" version of the file inside Fluent, which you can then modify. When you are ready to save your changes, you must keep the name and location exactly as it was when you opened it, otherwise Rocky will not be able to recognize the changes. When the changes are saved, you can click **Refresh** in Rocky to include them.

- If you wish to modify the mesh on the .cas file, save the modified .cas file in a new location outside of the project folder. Once the changes are saved, you need to load the newly saved .cas file inside the 2-way coupling option in Rocky's Data Editors panel.

- From inside Fluent, you can modify and save your original CAS (CAS.H5 or CAS.GZ) file. If you do this, you will have to restart the coupling process in Rocky by first selecting **No Coupling** from the CFD Coupling Mode list, and then following the steps to re-couple your Rocky simulation to the updated CAS (CAS.H5 or CAS.GZ) file.

ROCKY 2-WAY COUPLING FEATURES

Once your Fluent case is defined and you are ready to set up your 2-way coupled simulation in Rocky, there are several different options that can help you achieve your goals. Use the sections below to learn more.

ABOUT ADDITIONAL INPUT AND OUTPUT VARIABLES

Some external modules (p. 1391) might require certain variables or data to be exchanged with Fluent during the coupled simulation. In this version of Rocky, these additional input and output variables are applicable only with Fluent cases with **species transfer** enabled and which contains a mixture of various fluids.

For these special types of Fluent cases only, Rocky will display after Fluent case validation some additional variables on its **Variables** tab (Figures 1 and 12). On the **Variables** tab, **Additional Inputs** lists items that Fluent will receive from Rocky during the simulation, which Rocky instructs Fluent to create as User Defined Memories (UDMs). And **Additional Outputs** lists the items that Fluent will share during the simulation with Rocky.

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

Both types of additional variables can be viewed as Fluid properties in Rocky after processing, but note that they will not be calculated unless an external module is enabled that makes use of those variables. (Refer to your module's documentation for additional information.)

ABOUT SETTING OUTPUT FREQUENCY IN 2-WAY FLUENT COUPLING SIMULATIONS

In order for 2-Way Fluent simulations to be compatible with saving the project for tasks such as the three listed below, the **Simulation** Output Frequency setting on the **Solver** | **Time** tab is disabled and a new **Fluent Outputs Multiplier** parameter is provided. (See also About Solver Parameters (p. 464).)

- Saving a simulation for restart purposes (see also Save a Copy of a Partially Processed Simulation for Restart Purposes (p. 855))
- Resuming a simulation (Resume Processing a Stopped Simulation (p. 913))
- Extending the duration of a simulation (see also About Extending the Duration of an In-Process or Completed Simulation (p. 915))

Rocky uses the **Time Step Size** value that you set in Fluent to determine the initial Output Frequency of the Rocky and Fluent files. Rocky automatically adjusts its output frequency to match the time step set in the Fluent case. This means that by default, for every Fluent time step, one Rocky file and one Fluent file will be outputted.

You can change the frequency of outputs by using the **Fluent Outputs Multiplier**. A higher multiplier will result in fewer Rocky and Fluent files.

ABOUT FLUENT FILES AND STORAGE

In addition to the **Fluent Outputs Multiplier**, you are also given the option to **Keep all files** or determine how many of the last-saved DAT (and in some cases, as with moving meshes, CAS) **Files to keep**, which can help you manage your Fluent file storage.

Even if you increase your **Fluent Output Multiplier**, keeping all files is not always recommended for the 2-Way Fluent method due to the often large size of the Fluent mesh files. Unless you require the full set of files for post-processing in Fluent, it is recommended that you save only the last two (or more) Fluent files.

Important:

If you decide to save fewer than all of the files, be aware that you will not be able to post-process the unsaved files in Fluent. Also, be aware that the number of DAT (and sometimes CAS) files you save will be as far back in time as you can restart your simulation since fluid data will not be available for prior times.

Tip:

You can also choose which quantities and frequency you want to write to disk using Fluent's Autosave functionality.

ABOUT VOLUMETRIC DIFFUSION

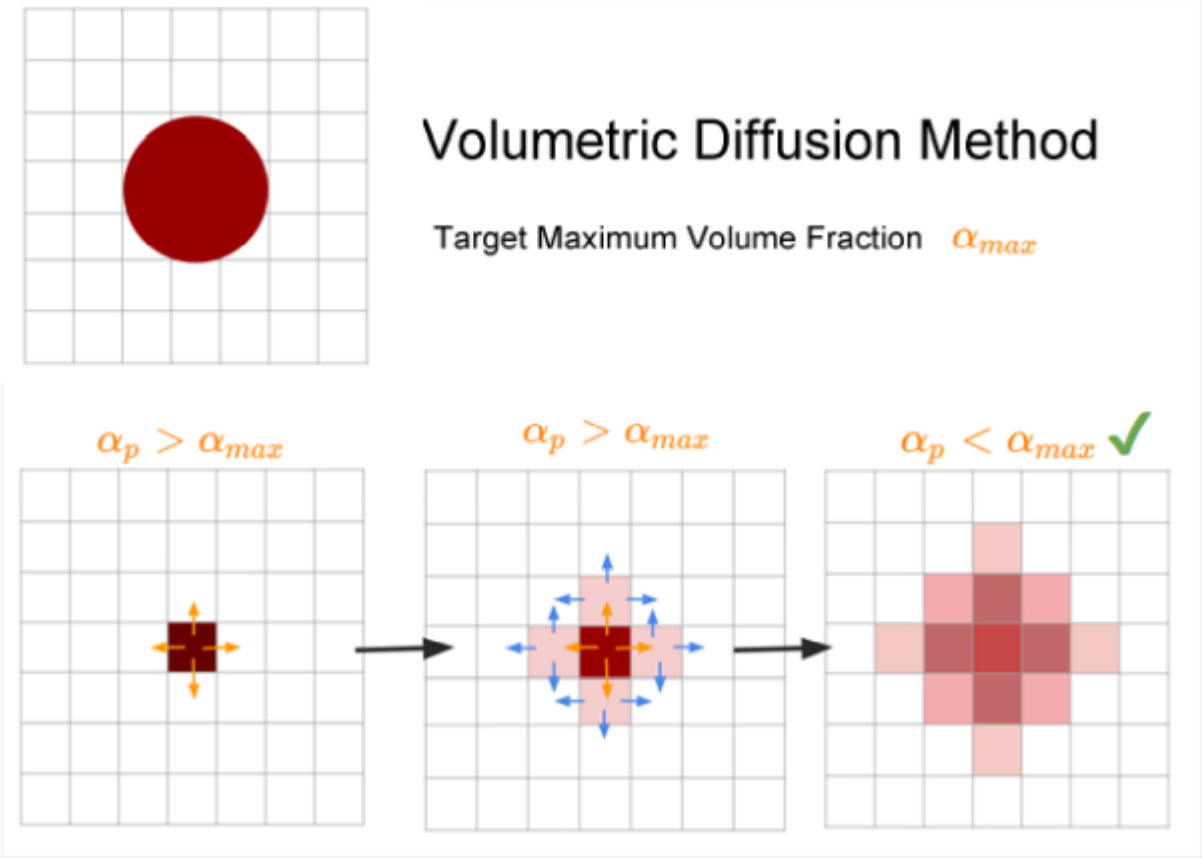
Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

If **Volumetric Diffusion** is selected for **Mapping Method**, then every iteration the volumetric diffusion equation is applied to the mesh. Rocky then decides if more iterations are required based upon the following criteria:

- Target Maximum Volume Fraction $\alpha_{\max}$
- Maximum Iterations N_{it}

See Figure 2 below for an illustration.

Figure 3.200: Volumetric Diffusion Method



Tip:

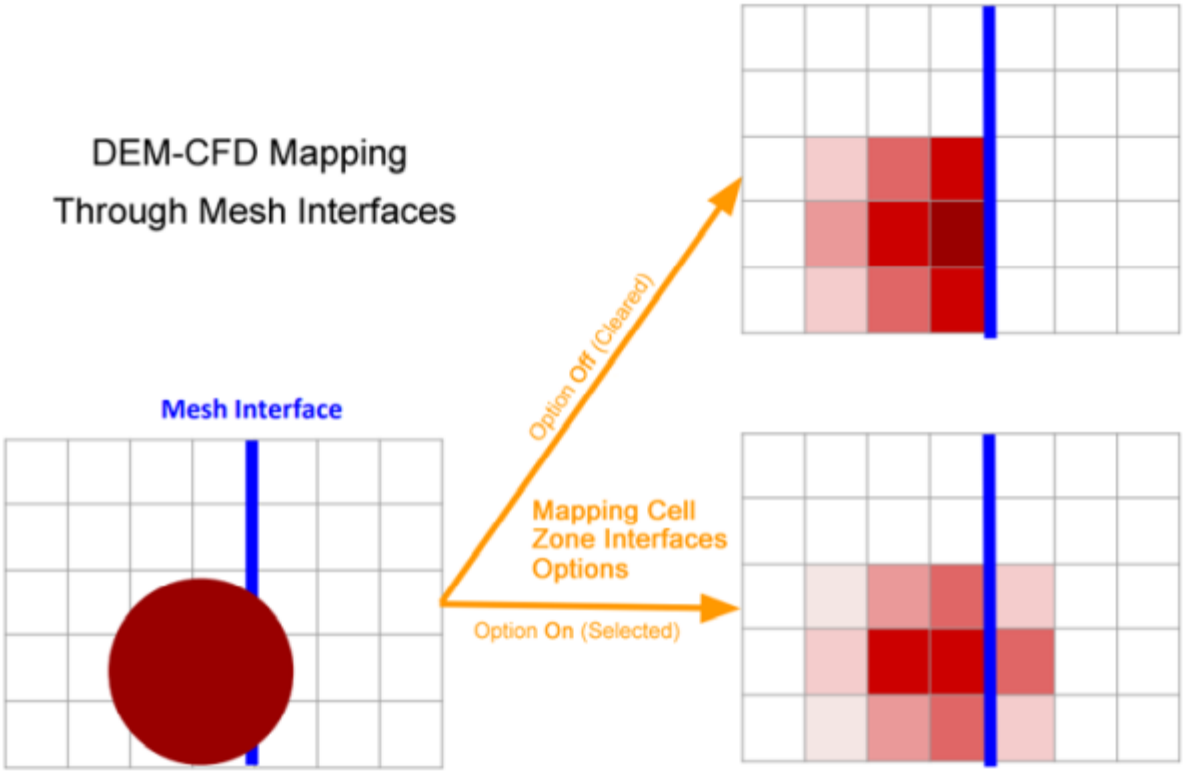
When Rocky's Advanced Features are enabled (see About Setting Global Preferences (p. 227)), these paraeters are available for tuning.

ABOUT MAPPING CELL ZONE INTERFACES

In this version of Rocky, you are able to select which interfaces you want to perform mapping across. This enables you to choose whether the averaging procedure will take into account CFD cells that are located on the opposite side of the interface or only on the same side of the interface as the cells (Figure 3).

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

Figure 3.201: Mapping Across Interfaces



For example, if the interface in question is not a physical boundary, and you created it because the case had sliding meshes or you simply decomposed the mesh in order to get a better quality, you may not want to block the transferring of the volume to the other side and in this case, would want to select the **Mapping Cell Zone Interfaces** option for that interface.

However, there may also be cases where you do not want to transfer the forces and volume fraction to the other side. For example, if in the CFD solver the interface is modeling a perforated plate or a grate, this should block particles from crossing the interface and therefore you can avoid the transferring of that volume by ensuring the **Mapping Cell Zone Interfaces** option is cleared for that interface.

Some best practices for using this method include:

- Avoid discontinuous volume fraction field and momentum and energy source terms near interfaces
- Avoid high volume fraction values

ABOUT COUPLED SIMULATIONS WITH "SHADOW BOUNDARIES"

A shadow boundary is an internal boundary without thickness. Due to the way Rocky optimizes the representation of the CFD domain in order to increase the simulation speed, in rare situations particles may be placed in a neighboring cell when the L-E mapping is performed. This is not a problem in the great majority of cases; however, when such a thing happens near a shadow boundary, there is a small chance that the particle is misplaced on the wrong side of the boundary. In order to avoid this chance occurrence, Rocky will prevent coupled simulations from processing if they have polygonal cells at shadow boundaries.

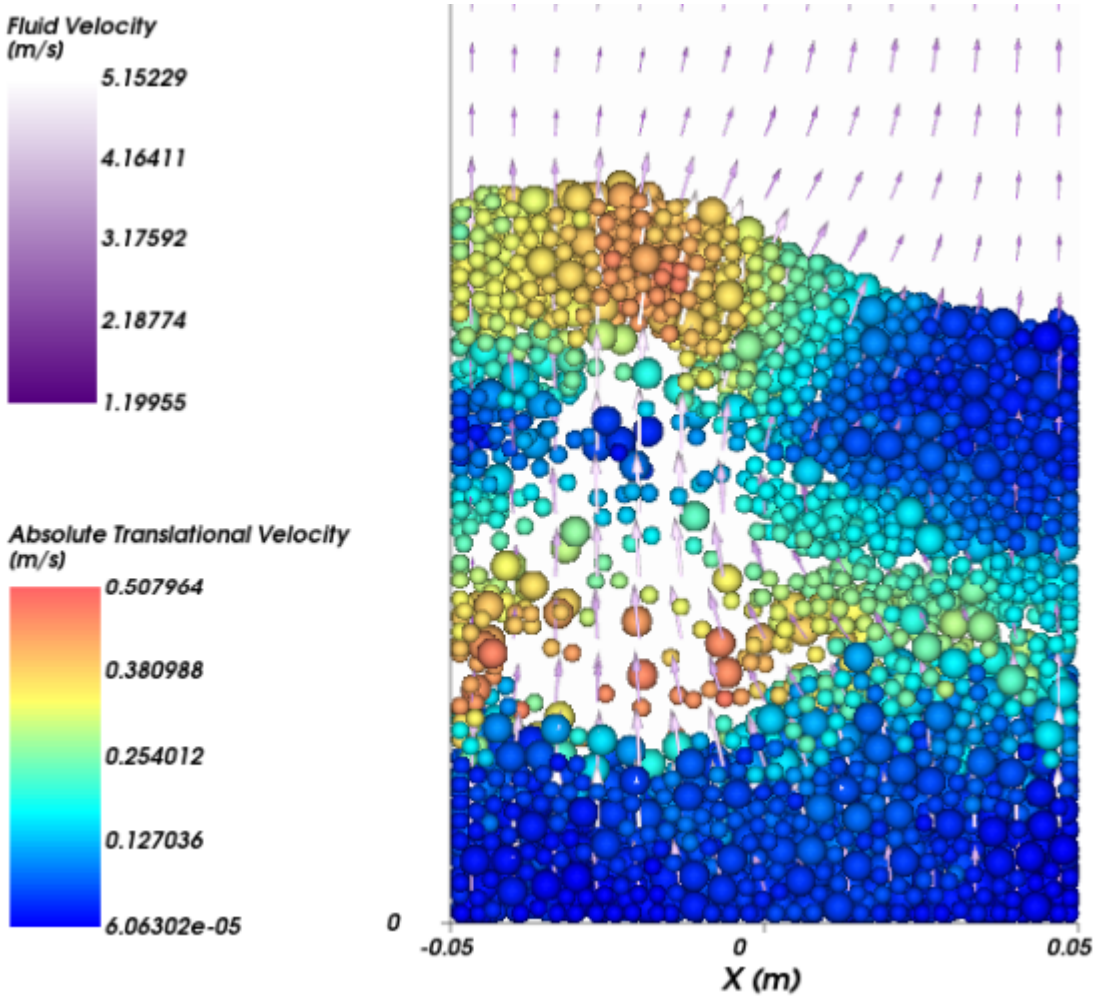
Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

To get around this limitation, add a thickness to your shadow boundaries, or avoid using polyhedron meshes around them.

ABOUT VISUALIZING 2-WAY FLUID AND PARTICLE FLOWS

In this version of Rocky, you are able to visualize in a 3D View window the transient fluid flow data represented by Nodes and/or Vectors. This display, when combined with Particle visualization, can provide you a fuller visual confirmation of your dynamic 2-Way coupled simulation (Figure 4).

Figure 3.202: Example of a 2-Way Fluent simulation showing transient fluid flow vectors (in purple), and transient particle nodes (all colors)



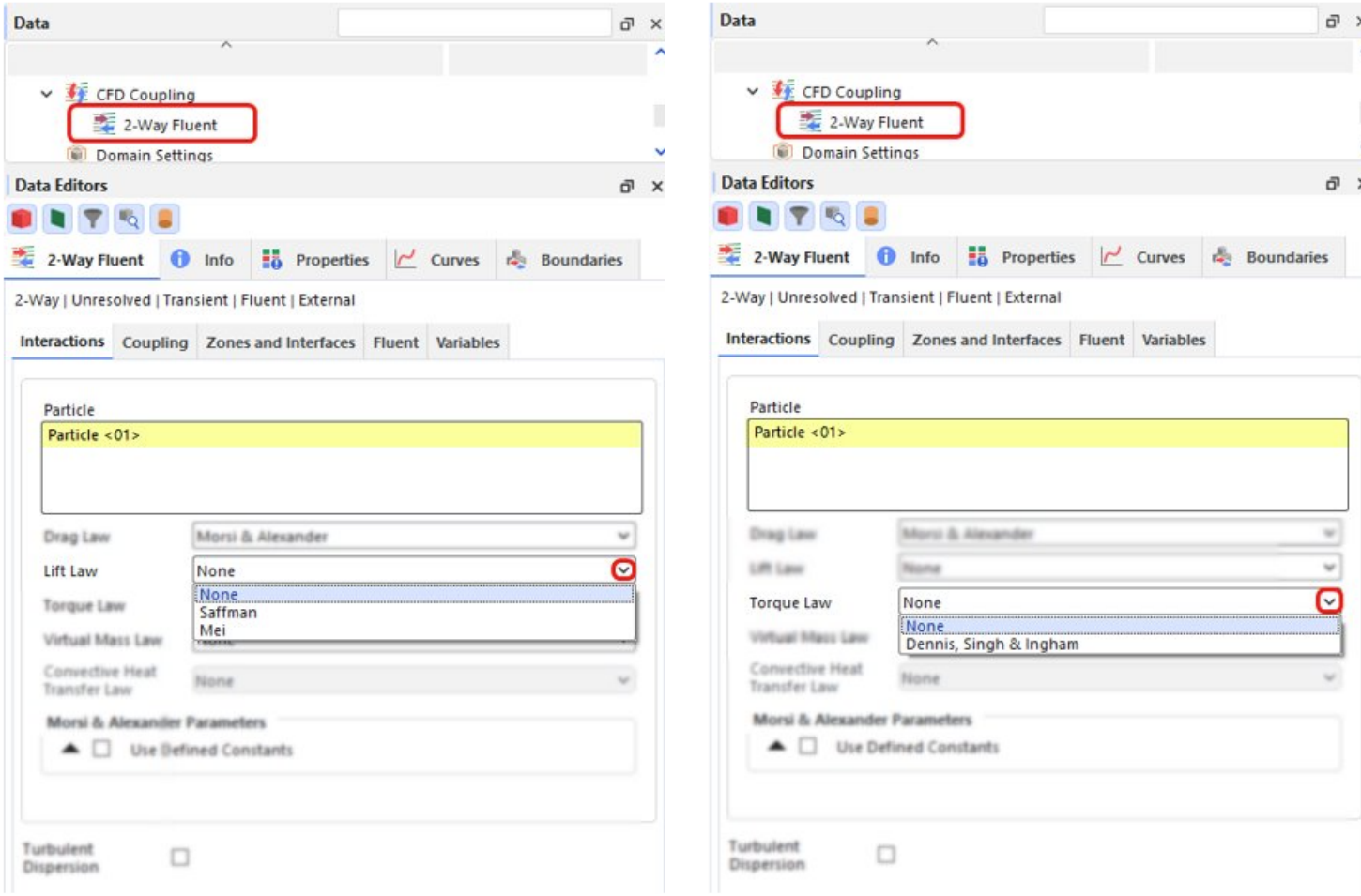
(See also About Using the Coloring Tab to Change a 3D View (p. 1074) and About Properties (p. 933).)

ABOUT LIFT AND TORQUE LAWS

In this version of Rocky, you are able to set torque and lift laws for Unresolved 2-way coupling with Fluent Figure 3.203: Lift and Torque Laws in the Data Editors panel (p. 805). All the current lift and torque laws are available, however they are suitable only for spherical particles and very dilute flows.

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

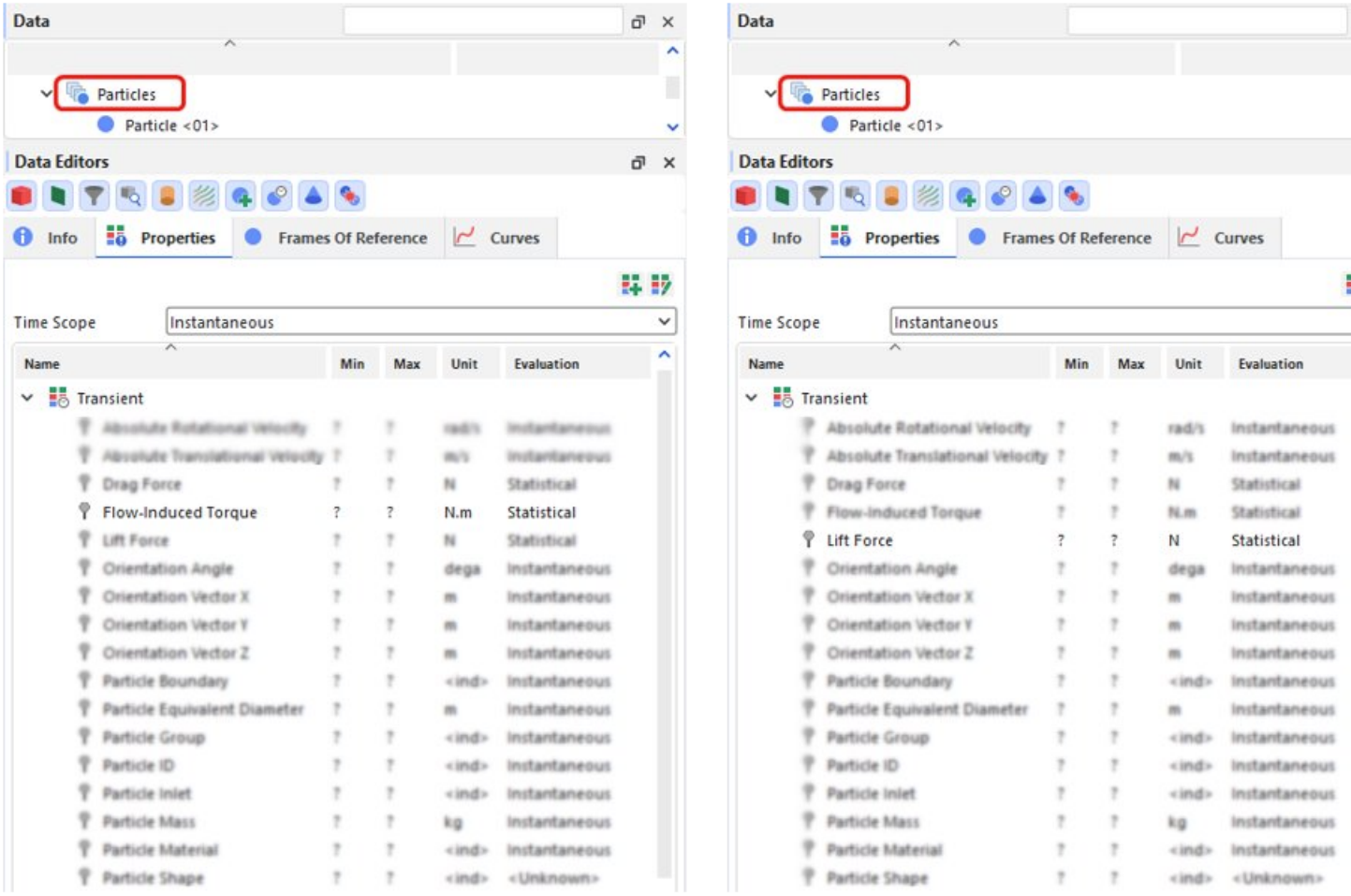
Figure 3.203: Lift and Torque Laws in the Data Editors panel



After processing your simulation, the referred properties will show up inside Particles in the Properties tab Figure 3.204: Lift and Torque Laws in the Data Editors panel after processing your simulation (p. 806).

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

Figure 3.204: Lift and Torque Laws in the Data Editors panel after processing your simulation



OTHER LIMITATIONS

Shell (p. 1395) and Solid (p. 1395) particles composed of Multiple Elements (also known as meshed (p. 1392) or flexible particles) are incompatible with 2-Way Fluent CFD Coupling. The only flexible particle shapes compatible with this feature are Fibers (p. 1391). (See also About Adding and Editing Particle Sets (p. 602).)

In addition, Shell (p. 1395) particles composed of Single Elements (also known as "rigid" particles) are also incompatible with 2-Way Fluent CFD Coupling.

Also, note that when combined with Coarse Grain models, only drag forces are considered. This means that other non-drag fluid force models, such as virtual mass and lift, are incompatible with Coarse Grain modeling.

For a full list of limitations, refer to the Ansys and CFD Coupling Limitations (p. 63) topic.

2-WAY FLUENT SETTINGS IN ROCKY

Use the images and tables below to help you understand how to set up the Rocky portion of the 2-Way Fluent method. Further details can be found in the following resources:

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

- CFD Coupling Technical Manual document (from the **Help** menu, point to **Manuals**, and then click **CFD Coupling Technical Manual**).
- Tutorial - Fluidized Bed in the *Rocky Tutorial Guide*

Figure 3.205: CFD Coupling, 2-Way Fluent, Interactions tab in the Data Editors panel

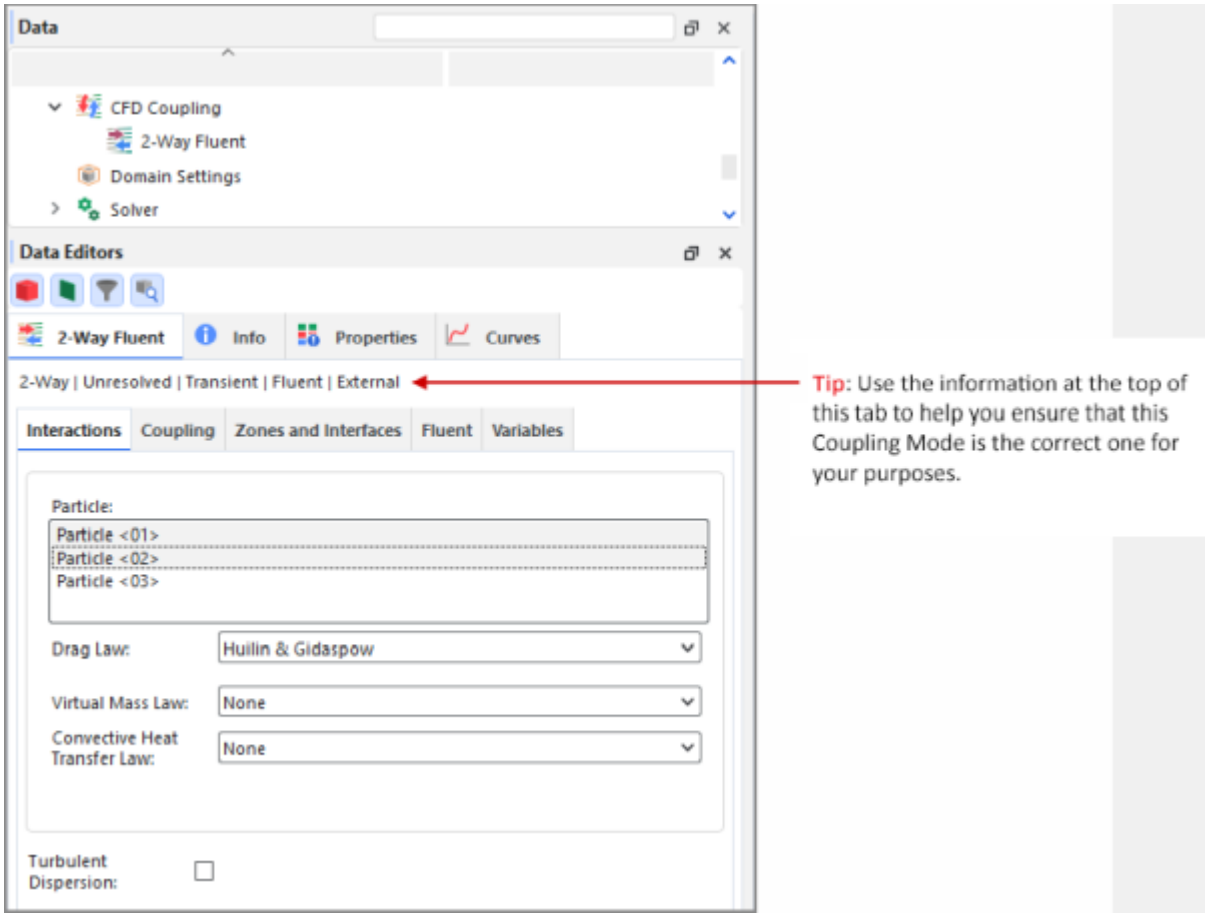


Table 1: 2-Way Fluent, Interactions Tab Settings

Setting	Description	Range
Particle	Permits correlations- such as drag, virtual mass, and convective heat transfer laws (if applicable)- to be defined per *Particle set* (p. 1393). You accomplish this by selecting (or multi-selecting) from a list of previously defined Particle sets, and then defining the applicable CFD Laws for only the selected set (or sets). **Important:** Even though they will still be listed here, neither flexible *Shell* (p. 1395) nor flexible *Solid* (p. 1395) particle shapes are supported in CFD Coupling approaches,	Automatically provided

Setting	Description	Range
	including this one. (See also Ansys and CFD Coupling Limitations (p. 63)).	
Drag Law	Correlation used by Rocky to calculate drag forces on particles. **Note:** With the exception of Ganser, which is incompatible with *Assembly* (p. 1389), *concave* (p. 1390) and *Shell* (p. 1395) particle shapes (see also About Adding and Editing Particle Sets (p. 602)), these drag laws apply to any supported particle shapes (see also Ansys and CFD Coupling Limitations (p. 63)) except *Fiber* (p. 1391) and also *Solid* (p. 1395) Sphero-Cylinder shapes, which have their own modified drag law based on the work by Marheineke and Wegener that should always be used for those shapes. **Tip:** For more information about these laws, refer to the CFD Coupling Technical Manual. (From the Rocky **Help** menu, point to **Manuals**, and then click **CFD Coupling Technical Manual**.)	Wen & Yu; Schiller & Naumann; Haider & Levenspiel; Ganser; Ergun; Gidaspow, Bezburuah & Ding; Huilin & Gidaspow; Di Felice; Dallavalle: Marheineke & Wegener; Syamlal & O'Brien; Morsi & Alexander; Hill, Koch & Ladd; Hill & Koch Note: If you have an *external Module* (p. 1391) that adds in an additional CFD law, you must use that law at least once in your simulation project if the Module is enabled. Refer to the Module's documentation (if provided) for more information. (See also Rocky Simulation Entities that can be Affected by Modules (p. 223)).
Lift Law	Correlation used by Rocky to calculate the fluid-particle force transversal to the direction of the relative velocity.	None; Saffman; Mei.
Torque Law	Correlation used by Rocky to calculate the torque exerted by the fluid over the particle.	None; Dennis, Singh and Ingham.
Virtual Mass Law	Correlation used by Rocky to calculate the virtual mass coefficient. **Tip:** For more information about these laws, refer to the CFD Coupling Technical Manual. (From the Rocky **Help** menu, point to **Manuals**, and then click **CFD Coupling Technical Manual**.)	None; Constant; Paladino; Ishii & Mishima Note: If you have an *external Module* (p. 1391) that adds in an additional CFD law, you must use that law at least once in your simulation project if the Module is enabled. Refer to the Module's

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

Setting	Description	Range
		documentation (if provided) for more information. (See also Rocky Simulation Entities that can be Affected by Modules (p. 223)).
Convective Heat Transfer Law	When both **Thermal Model** is enabled (see also About Physics Parameters (p. 417)) and the selected Fluent CAS (CAS.H5 or CAS.GZ) file is set to solve energy equations, this correlation is used by Rocky to calculate the fluid-particle convective heat transfer coefficient. **Note:** Note: The thermal (energy) solution option must be the same in both Fluent and Rocky setups.	None; Ranz & Marshall; Whitaker; Gunn Note: If you have an *external Module* (p. 1391) that adds in an additional CFD law, you must use that law at least once in your simulation project if the Module is enabled. Refer to the Module's documentation (if provided) for more information. (See also Rocky Simulation Entities that can be Affected by Modules (p. 223)).
Use Defined Constants	When **Morsi & Alexander** is selected for **Drag Law**, this enables you to determine how the model's three coefficients will be determined. Specifically: • When cleared, the standard values provided with the model will be used. See the *CFD Coupling Technical Manual* for more detail. (From the Rocky **Help** menu, point to **Manuals**, and then click *CFD Coupling Technical Manual*.) • When enabled, you can define your own set of coefficients that will be used by the model.	Turns on or off
K_1	When **Use Defined Constants** is enabled, this is the value you want assigned to the K_1 constant used in the **Morsi & Alexander** drag law calculation. See also the *CFD Coupling Technical Manual*. (From the R ocky **Help** menu, point to **Manuals**, and then click *CFD Coupling Technical Manual*.)	Any value
K_2	When **Use Defined Constants** is enabled, this is the value you want assigned to the K_2 constant used in the **Morsi & Alexander** drag law calculation. See also the *CFD Coupling Technical Manual*. (From the R ocky **Help** menu, point to **Manuals**, and then click *CFD Coupling Technical Manual*.)	Any value

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

Setting	Description	Range
K_3	When **Use Defined Constants** is enabled, this is the value you want assigned to the K_3 constant used in the **Morsi & Alexander** drag law calculation. See also the *CFD Coupling Technical Manual*. (From the R ocky **Help** menu, point to **Manuals**, and then click *CFD Coupling Technical Manual*.)	Any value
Syamlal & O'Brien Parameters c1	When **Syamlal & O'Brien** is selected for **Drag Law**, this defines the c1 coefficient required by the drag law. Tip:For more information about this parameter, refer to the CFD Coupling Technical Manual. (From the Rocky **Help** menu, point to **Manuals**, and then click **CFD Coupling Technical Manual**.)	Any value
Syamlal & O'Brien Parameters d1	When **Syamlal & O'Brien** is selected for **Drag Law**, this defines the d1 coefficient required by the drag law. Tip:For more information about this parameter, refer to the CFD Coupling Technical Manual. (From the Rocky **Help** menu, point to **Manuals**, and then click **CFD Coupling Technical Manual**.)	Any value
Turbulent Dispersion	When the imported CAS (CAS.H5 or CAS.GZ) file has a turbulent model that uses turbulence kinetic energy as one of its quantities, this allows those turbulent parameters to be used and modified in Rocky. Clearing this checkbox prevents the turbulent parameters from being used during the Rocky simulation. **Tip:** For more information about this parameter, refer to the CFD Coupling Technical Manual. (From the Rocky **Help** menu, point to **Manuals**, and then click **CFD Coupling Technical Manual**.)	Turns on or off

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

Figure 3.206: CFD Coupling, 2-Way Fluent, Coupling tab in the Data Editors panel

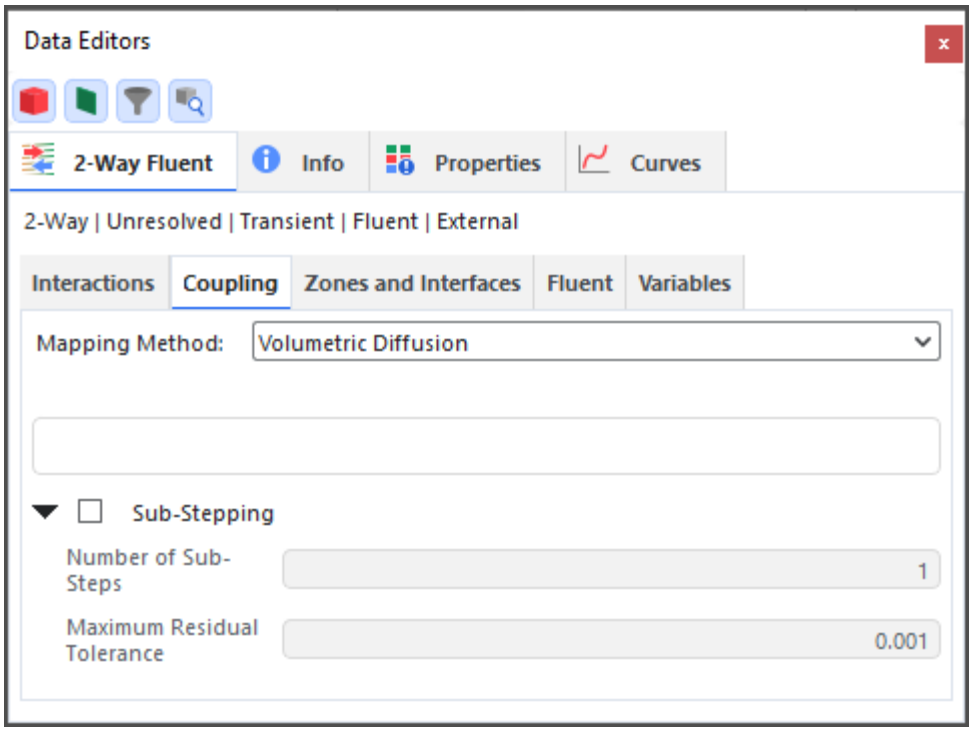
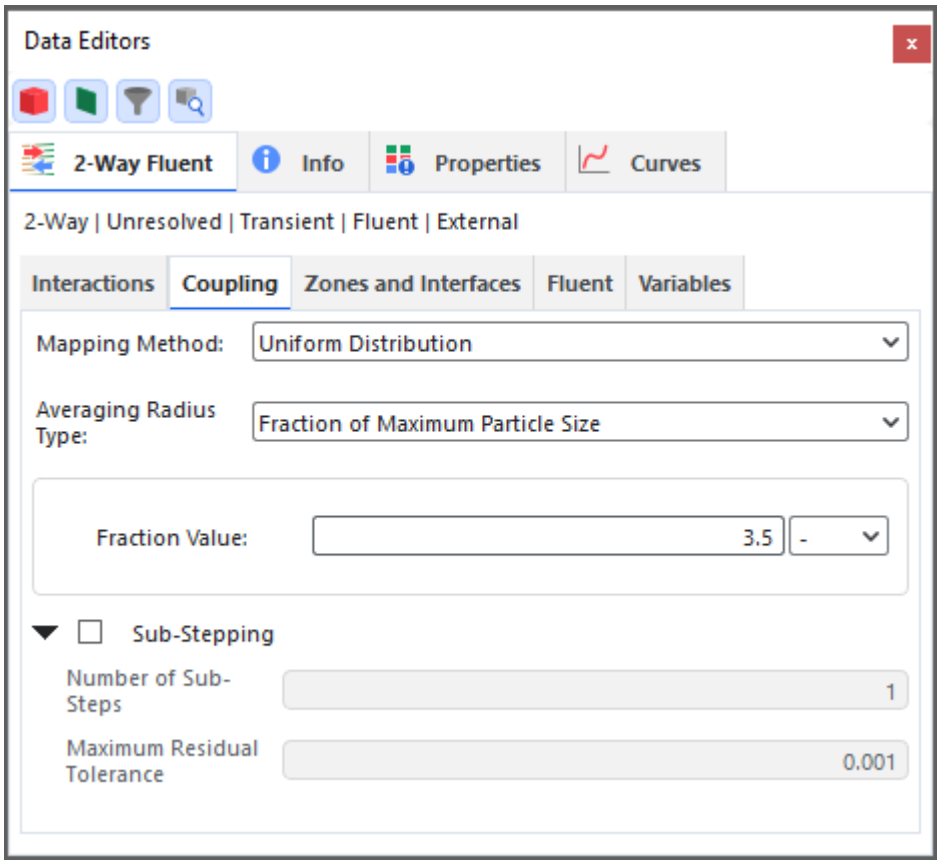


Figure 3.207: CFD Coupling, 2-Way Fluent, Coupling tab when Uniform Distribution is selected for Mapping Method



Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

Table 2: 2-Way Fluent, Coupling Tab Settings

Setting	Description	Range
Mapping Method	Helps facilitate smoother Fluent calculations by distributing each particle's information across a specific mesh region. These regions are composed of the Fluent mesh cell where the particle is located and its neighboring cells. The two options for employing this mapping method are as follows: • Uniform Distribution : The average of each particle's information is calculated uniformly between all the cells contained within the specific region. In this situation, all the cells involved will receive the same quantity of information. • **Volumetric Diffusion**: The average of each particle's information is calculated by iterating a diffusion algorithm to the mesh. On each iteration, a calculated amount of exceeding particle volume is exchanged between cells and their immediate neighbors, together with calculated amounts of other particle's information. These amounts are functions of the difference of values between the cell and its neighbors, and are tuned by automatic detection of individual optimal diffusion coefficients for each cell. Iterations are repeated until a target volume fraction is achieved for the mesh (or a maximum number of iterations is reached). When Rocky's Advanced Features are enabled (see About Setting Global Preferences (p. 227)), the following parameters and default values are available for tuning this mapping method: – **Maximum Iterations**: 500 – **Solids Maximum Volume Fraction Target**: 0.65 • **Diffusion Equation (Beta)**: The diffusion solution mapping process solves a diffusion equation to perform the mapping and smoothing of the volume fraction variables (particle volume fraction and particle volume fraction variation due to particles entering and leaving the domain) and a pure advection equation to all other quantities. This way all other mapped quantities (averaged particle quantities, momentum, and energy sources, for example) are all distributed according to the volume fraction distribution. The Back-Diffusion method uses the same approach as the pure-advection of the DEM quantities applied in the Diffusion Solution mapping scheme, but uses the inverse of the volume fraction diffusion velocity as the advection velocity, to compute a volume fraction weighted transport of the fluid properties and flow quantities at the position where the particles are located. (See also Figure 1 above.)	Uniform Distribution; Volumetric Diffusion

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

Setting	Description	Range
Averaging Radius Type	When **Uniform Distribution** is selected for **Mapping Method**, this determines what radius around a Fluent mesh cell Rocky will average the neighboring cells' particle information. This step is necessary to ensure smooth calculations for Fluent. The higher the values chosen, the more effective the smoothing but the more time to process. Specifically, choose: • **Fraction of Maximum Particle Size** when you want the radius to be equal to a fraction amount of your choosing multiplied by the maximum particle size. • **Absolute Value** when you want to set the radius value yourself. It is recommended that whatever value you choose includes 1-2 additional Fluent mesh cells only.	Fraction of Maximum Particle Size; Absolute Value
Fraction Value	When **Averaging Radius Type** is set to **Fraction of Maximum Particle Size**, this value will be multiplied by the maximum particle size to achieve the averaging radius.	Any value
Absolute Value	When **Averaging Radius Type** is set to **Absolute Value**, this is value will be used as the averaging radius.	Any value
Maximum Iterations	When the **Advanced Features** checkbox is enabled on the **Options | Preferences** dialog (see also About Setting Global Preferences (p. 227)), and **Volumetric Diffusion** is selected for **Mapping Method**, the mapping process is interrupted if the Maximum Volume Fraction Target is not achieved during this number of iterations. This parameter has lower precedence than Minimum Iterations above. **Note:** This text field does not support parametric variables. (See also I cannot enter an input variable or mathematical function into a text field (p. 1334)).	Values 10-1000 **Tip:** The recommended value is 500 **Note:** Small values of this parameter may result in Rocky not achieving the

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

Setting	Description	Range
		Maximum Volume Target Fraction you have specified.
Solids Maximum Volume Fraction Target	When the **Advanced Features** checkbox is enabled on the **Options | Preferences** dialog (see also About Setting Global Preferences (p. 227)), and **Volumetric Diffusion** is selected for **Mapping Method**, this defines the maximum volume fraction—of only the solids (particles); the fluid is not considered—to be honored during the simulation.	Values greater than 0 (zero) but less than 1 **Tip:** The recommended values are between 0.6 and 0.85
Sub-Stepping	Allows you to choose how often during processing Rocky will interact with Fluent inside a single Fluent time step. Enabling Rocky to interact more frequently with Fluent increases stability and accuracy of the coupled calculations, but can increase the processing time. To counteract these effects, you can combine sub-stepping with a higher Fluent time step to achieve a better balance between processing speed and stability/accuracy. When cleared, Rocky will interact only once with Fluent during each Fluent time step. If enabled, Rocky will interact more than once with Fluent during each Fluent time step.	Turns on or off
Number of Sub-Steps	When **Sub-Stepping** is enabled, this defines the number of times Rocky will interact with Fluent during each Fluent time step.	2 - 50
Maximum Residual Tolerance	When **Sub-stepping** is enabled, Rocky overwrites the residual tolerances defined in the Fluent CAS (CAS.H5 or CAS.GZ) file with this value, which defines the largest allowable value of residual error during each interaction Rocky has with Fluent. The solution is obtained when the residual error during an iteration in any computational node is less than the specified tolerance.	1e-16 - 1.0 **Important:** Setting this value enables

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

Setting	Description	Range
		Rocky to overwrite the residual tolerances defined in the Fluent CAS (CAS.H5 or CAS.GZ) file.
Overwrite CFD Update Distance	When the **Advanced Features** checkbox is enabled on the **Options | Preferences** dialog (see also About Setting Global Preferences (p. 227)), this allows you to overwrite the default distance used as a criteria for updating the volume fraction field. As this step may take a considerable amount of time, increasing the associated **CFD Update Distance** value (and therefore reducing the frequency at which the volume fraction is updated) might help reduce processing time.	Turns on or off
CFD Update Distance	When the **Advanced Features** checkbox is enabled on the **Options | Preferences** dialog (see also About Setting Global Preferences (p. 227)), and the **Overwrite CFD Update Distance** checkbox is enabled, this defines the maximum distance the particles are allowed to travel before a volume fraction field update. The smaller the value, the more frequent the update. As this step may take a considerable amount of time, increasing the update distance (and therefore reducing the frequency at which the volume fraction is updated) might help reduce processing time.	Positive values

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

Figure 3.208: 2-Way Fluent, Zones and Interfaces tab

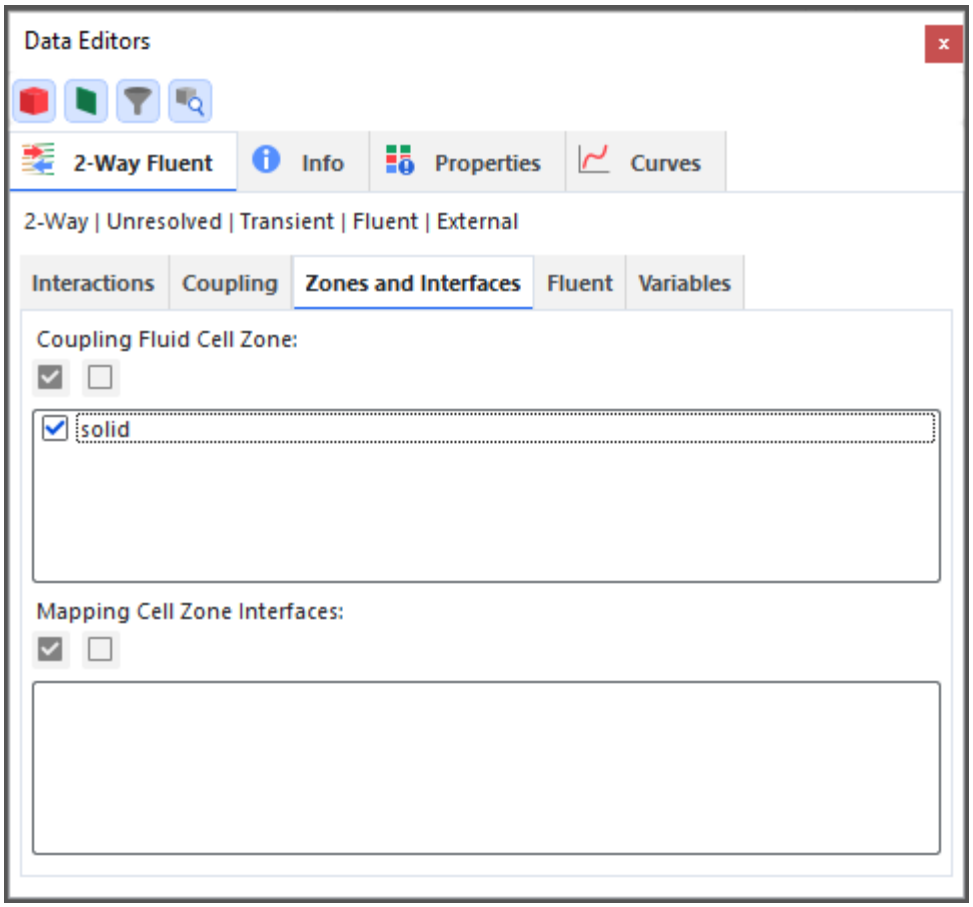


Table 3: 2-Way Fluent, Zones and Interfaces Tab Settings

Setting	Description	Range
Coupling Fluid Cell Zone	Enables you to choose which fluid cell zones you want included in the coupled calculations. Note: At least one fluid zone must be selected in order to process the coupled simulation.	Turns on or off
Mapping Cell Zone Interfaces	Enables you to select which interfaces you want to perform the mapping across. If selected, the averaging procedure will take into account CFD cells that are located on the other side of the interface, whereas if not selected, the mapping will take into account only cells that are on the same side of the interface as the cell. (See also Figure 2 above.)	Turns on or off

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

Figure 3.209: CFD Coupling, 2-Way Fluent, Fluent tab in the Data Editors panel

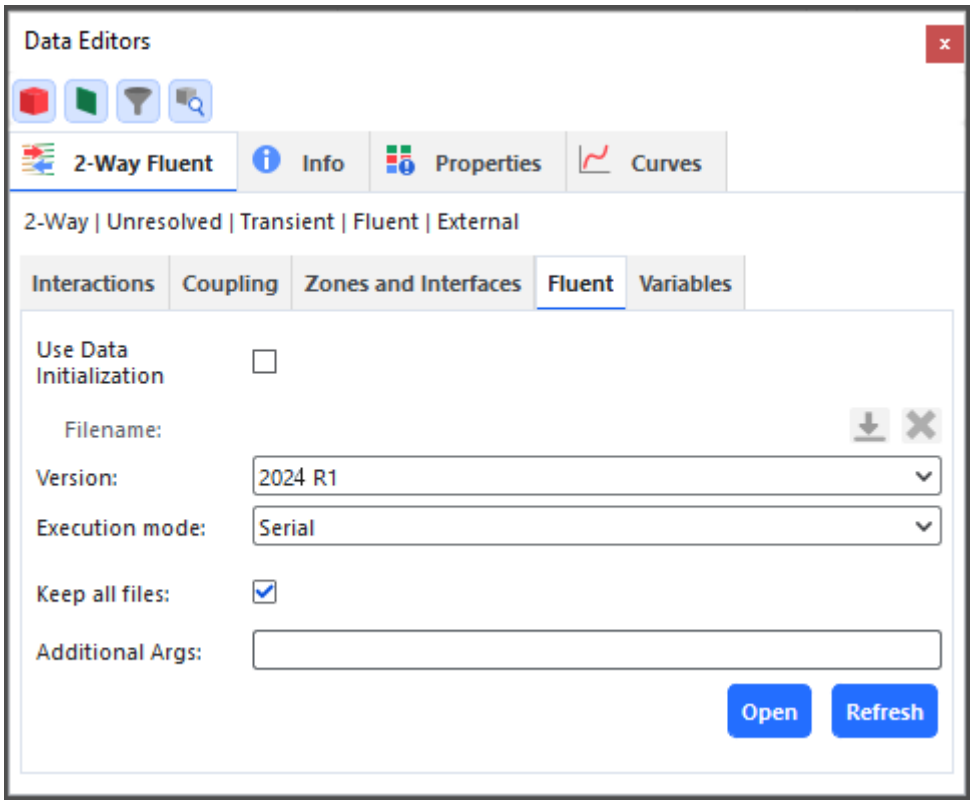
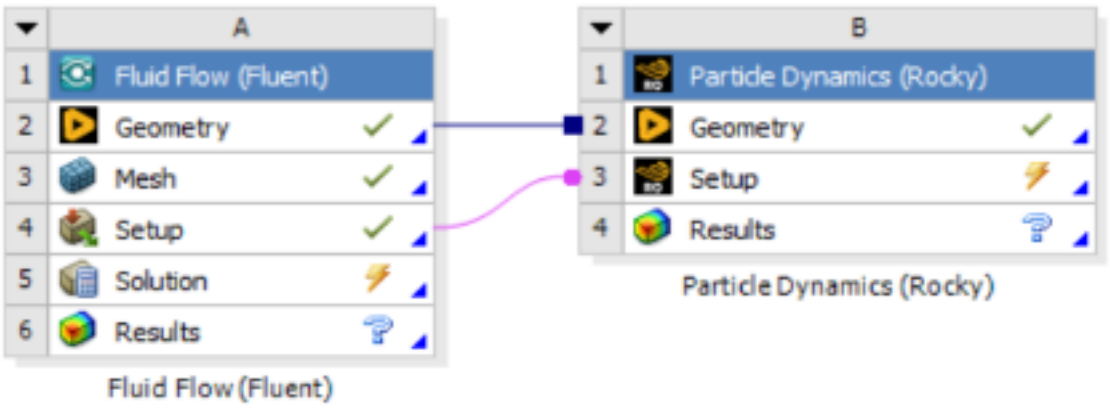


Figure 3.210: 2-Way Fluent, Fluent tab when Local Parallel is selected for Execution mode



Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

Figure 3.211: 2-Way Fluent, Fluent tab when Distributed Parallel is selected for Execution mode

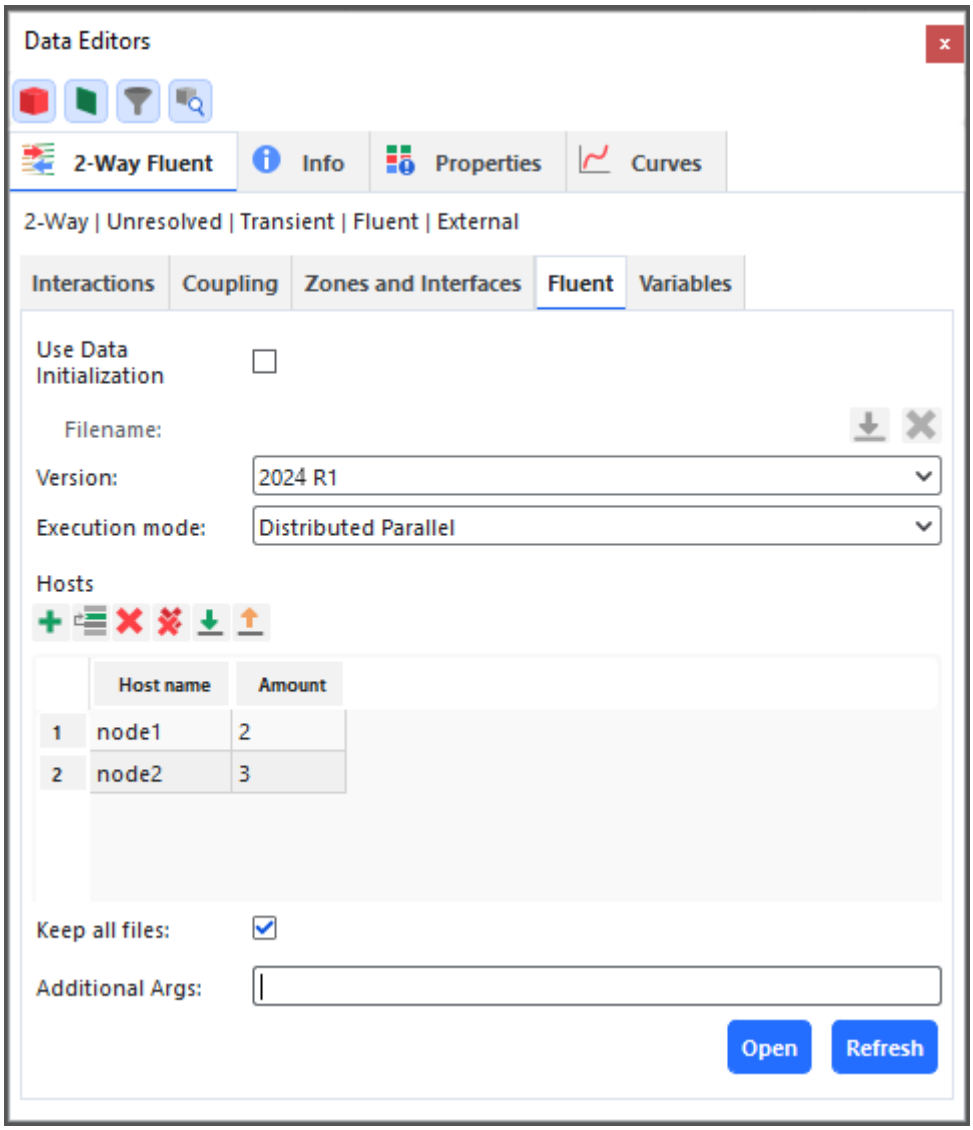


Table 4: 2-Way Fluent, Fluent Tab Settings

Setting	Description	Range
Rocky Phase	For *multiphase* (p. 1392) CAS (CAS.H5 orCAS.GZ) files only, this enables you to choose which secondary phase of the Fluent file represents particle flow information. **Note:** *single phase* (p. 1395) CFD files will not list this option as the Rocky Phase can be automatically determined.	List automatically generated based upon what secondary phases were entered in the Fluent CAS (CAS.H5 or CAS.GZ) file.

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

Setting	Description	Range
Use Data Initialization	When selected, enables you to begin your simulation with an initial flow field based upon the Fluent fluid phases. This initial solution is saved from Fluent into a DAT (DAT.H5 or DAT.GZ) file that is then used by Rocky during processing. **Important:** If you choose to use initial fluid flow data in your simulation, it is important that the secondary phase (particulate phase) has a zero volume fraction. This is calculated in Rocky and sent to Fluent.	Turns on or off
Version	Enables you to select the Ansys Fluent software version you want to use for processing. **Tip:** If you do not see the Ansys version you expected, see also Rocky does not list my Ansys version when I try to set up 2-Way Fluent Coupling (p. 1326).	List automatically generated based upon the software versions detected.
Execution mode	Enables you to select how your computer and/or network will process the fluid portion of the coupled simulation. Note: Regardless of what is selected here, the Rocky portion of the coupled simulation will still be processed according to the **Processing Unit** and related information that you specified on **Solver** | **General Settings** tab (see also About Starting a Simulation (p. 908). Specifically: • **Serial** allows a single Fluent process on your local computer to calculate the fluid portion of the coupled simulation. • **Local Parallel** allows you to select multiple Fluent processes on your local computer from which to calculate the fluid portion of the coupled simulation. • **Distributed Parallel** allows you to spread the fluid processing portion of the coupled simulation across multiple separate machines on your network.	Serial; Local Parallel; Distributed Parallel

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

Setting	Description	Range
Solver Processes	When **Local Parallel** is selected for **Execution mode**, this enables you to determine the number of computer processors (threads) you want dedicated to the CFD solver. **Note:** You select the number of processors used by Rocky when you start processing the simulation. (See also About Starting a Simulation (p. 908).) **Note:** This text field does not support parametric variables. (See also I cannot enter an input variable or mathematical function into a text field (p. 1334).)	Positive values
Host Name	When **Distributed Parallel** is selected for **Execution mode**, this is the name of the remote machine on your network. Tip:Contact your IT administrator to find out the host names of the fluid nodes on your network.	99 character limit
Amount	When **Distributed Parallel** is selected for **Execution mode**, for each separate **Host Name** listed, this is the number of computer processors you want dedicated to the CFD solver.	Positive values
Keep all files	When selected, all Fluent DAT (DAT.H5 or DAT.GZ) files (and in some cases, as with moving meshes, CAS files as well) will be saved every time a Rocky output file is saved. When cleared, this enables you to select how many DAT (and sometimes CAS) **Files to keep**. Keeping fewer than all the DAT (and sometimes CAS) files can help you save storage space when processing your simulation. However, doing so can also limit your fluid post-processing abilities, as well as your ability to restart the simulation from an earlier time due to the reduced amount of past fluid data that is available when you save fewer DAT (and sometimes CAS) files.	Turns on or off
Files to keep	When **Keep all files** is cleared, this determines how many Fluent DAT (DAT.H5 or DAT.GZ) files (and in some cases, as with moving meshes, CAS files as well) are saved with each Rocky output file saved. For example, if set to 1, only the last DAT (and sometimes CAS) file will be kept. If set to 2, only the last 2 DAT (and sometimes CAS) files will be kept, and so on.	Whole values greater than or equal to 1

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

Setting	Description	Range
	This setting can be useful in cases where you need only to keep files past steady state, for example, or when you do not require the Fluent files for post-processing. **Important:** If you decide to save fewer than all of the files, the number of DAT (and sometimes CAS) files you save will be as far back in time as you can restart your simulation since fluid data will not be available for prior times. **Note:** This text field does not support parametric variables. (See also I cannot enter an input variable or mathematical function into a text field (p. 1334).)	
Additional Args	This displays the set of arguments that Rocky passed "as-is" from the Fluent command line when the coupled simulation began. For advanced users, this enables more control of the Fluent portion of the processing. For more information about these arguments, see your Ansys Fluent documentation.	Positive values

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

Figure 3.212: CFD Coupling, 2-Way Fluent, Variables tab in the Data Editors panel

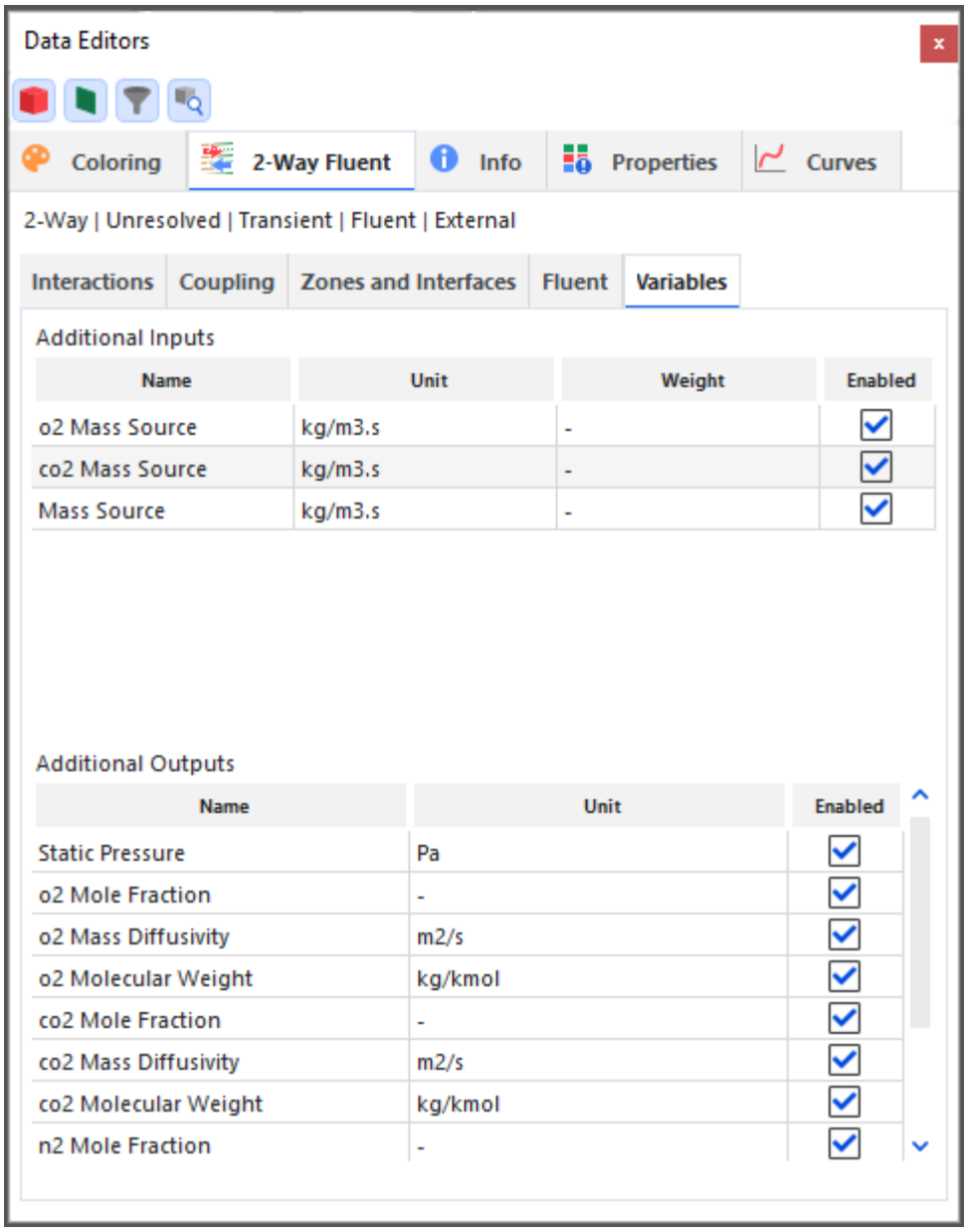


Table 5: 2-Way Fluent, Variables Tab Display

Property	Display Description
Additional Inputs	
Name	The name of the additional variable or data that Fluent will receive from Rocky during the simulation. Each **Additional Input** has a corresponding UDM in Fluent.
Unit	The SI unit of the variable.
Weight	The name of the variable that will be used as a weight for the selected averaging method. This variable can be another **Additional Input** or

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

Property	Display Description
	Additional Output variable, or can be a standard variable in Rocky, such as Fluid Velocity.
Additional Outputs	
Name	The name of the additional variable or data that Fluent will share with Rocky during the simulation.
Unit	The SI unit of the variable.

What would you like to do?

- Exchange Rocky Particle Data with Fluid Flow Data from Ansys Fluent (p. 827), which covers only the final coupling portion of the entire 2-Way Fluent Coupling procedure.
- Use the 2-Way Fluent Method to Process a Fluent and Rocky Simulation (p. 386), which covers the full procedure, including the setup steps that are required in Fluent.

See Also

- Set or Modify Fluid and/or Air Flow Properties (p. 757)
- About Using the Coloring Tab to Change a 3D View (p. 1074)
- I cannot find Particle Size Scale Factor (p. 1330)

How to Set initial Fluid Field to 2-way coupled simulations in Workbench

Before starting the 2-way coupled run, you must initialize the CFD flow field in the entire CFD domain. When setting up a 2-way coupling simulation between Rocky and Fluent in Workbench, there are two options to set initial values for the flow variables, and initialize the solution using these values:

1.Initialize with values from the case.

The values set in the Solution Initialization task page from Fluent will be used to generate the initial fluid flow field. These fluid values can be computed from information in a specified zone, entered manually, or computed based on all zones by the solver.

2.Use existing solution data.

The desired information is automatically imported from the Fluent case that already run and used as the initial flow field for the coupled run.

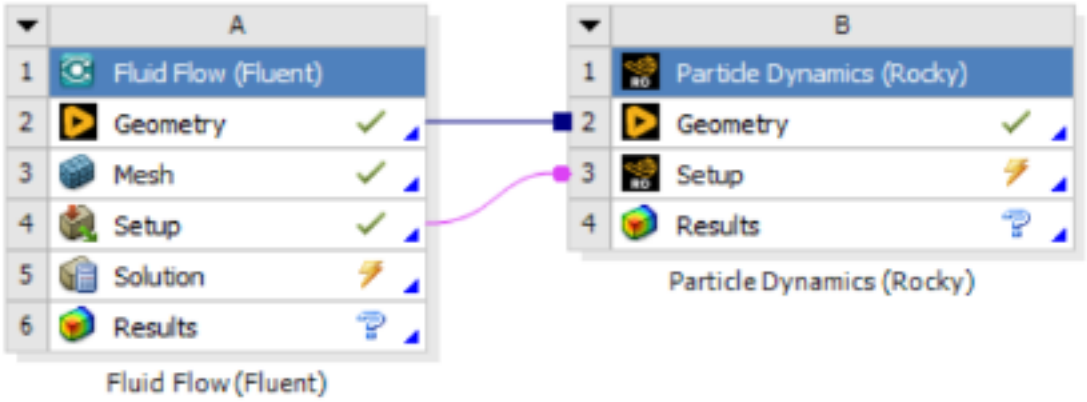
You can use this option to facilitate convergence of the coupled simulation by providing a converged flow field without particles, or provide different values for particular variables, such as temperature or phase volume fraction, into different cells using the Patch option in Fluent.

HOW TO INITIALIZE WITH VALUES FROM THE CASE

The coupled simulation will initialize the fluid field using the values set in the Solution Initialization task page from Fluent if only the Setup entry from the Fluid Flow Field is linked to the Particle Dynamics Setup.

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

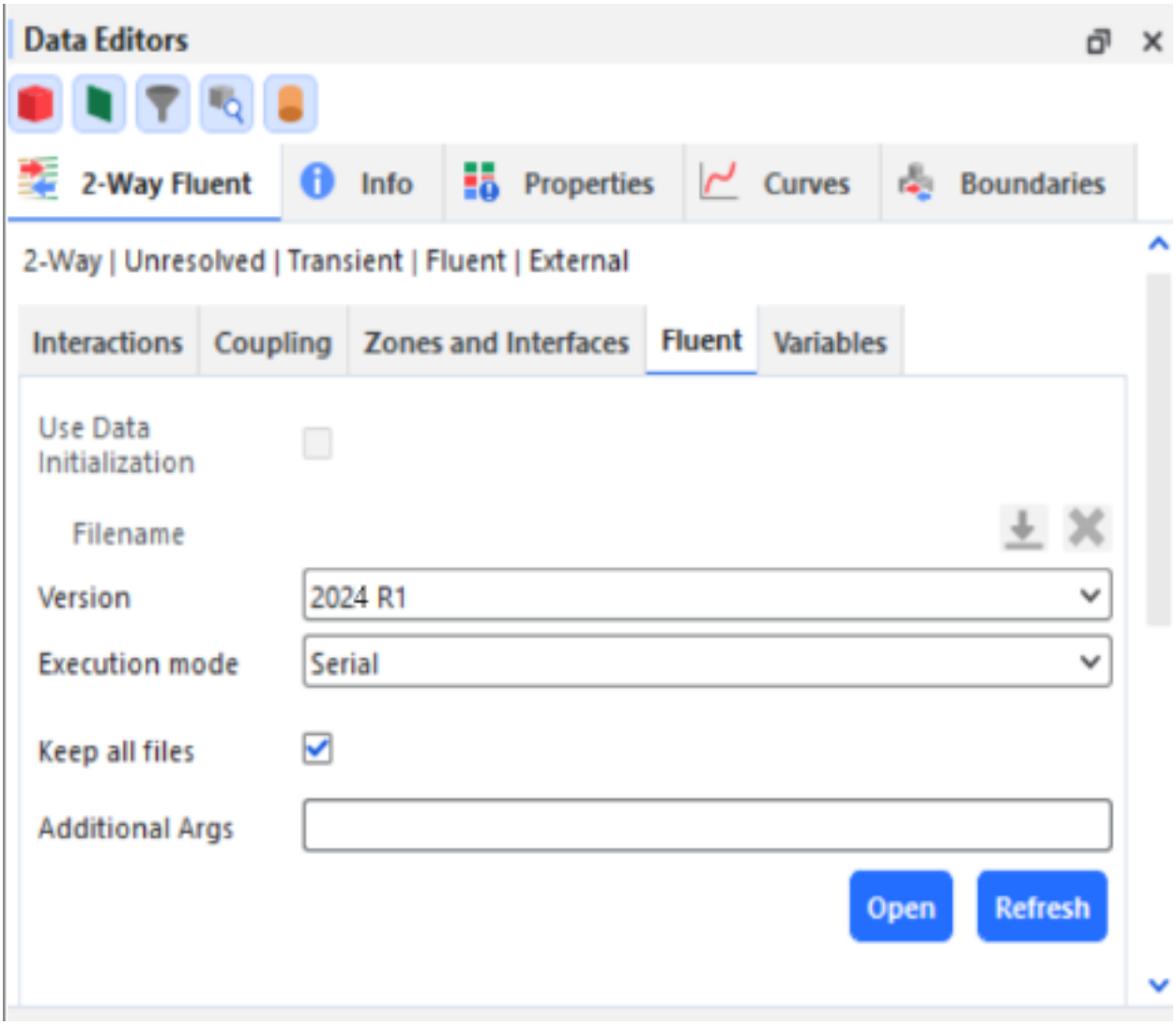
Figure 3.213: 2-way Coupled Rocky-Fluent simulation in Workbench initializing the fluid field using the values set in the Fluent setup



The Fluent tab of the 2-Way Fluent editor will have the option Use Data Initialization unchecked and blocked.

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

Figure 3.214: Fluent tab in Rocky Data Editors when coupling within workbench without initial fluid data file

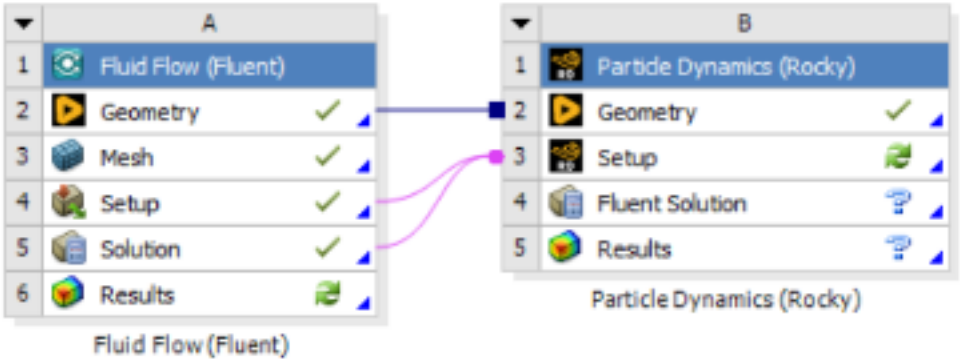


HOW TO INITIALIZE WITH EXISTING FLUID DATA

The coupled simulation will use the existing fluid data if the Setup and Solution entries from the Fluid Flow Field are linked to the Particle Dynamics Setup.

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

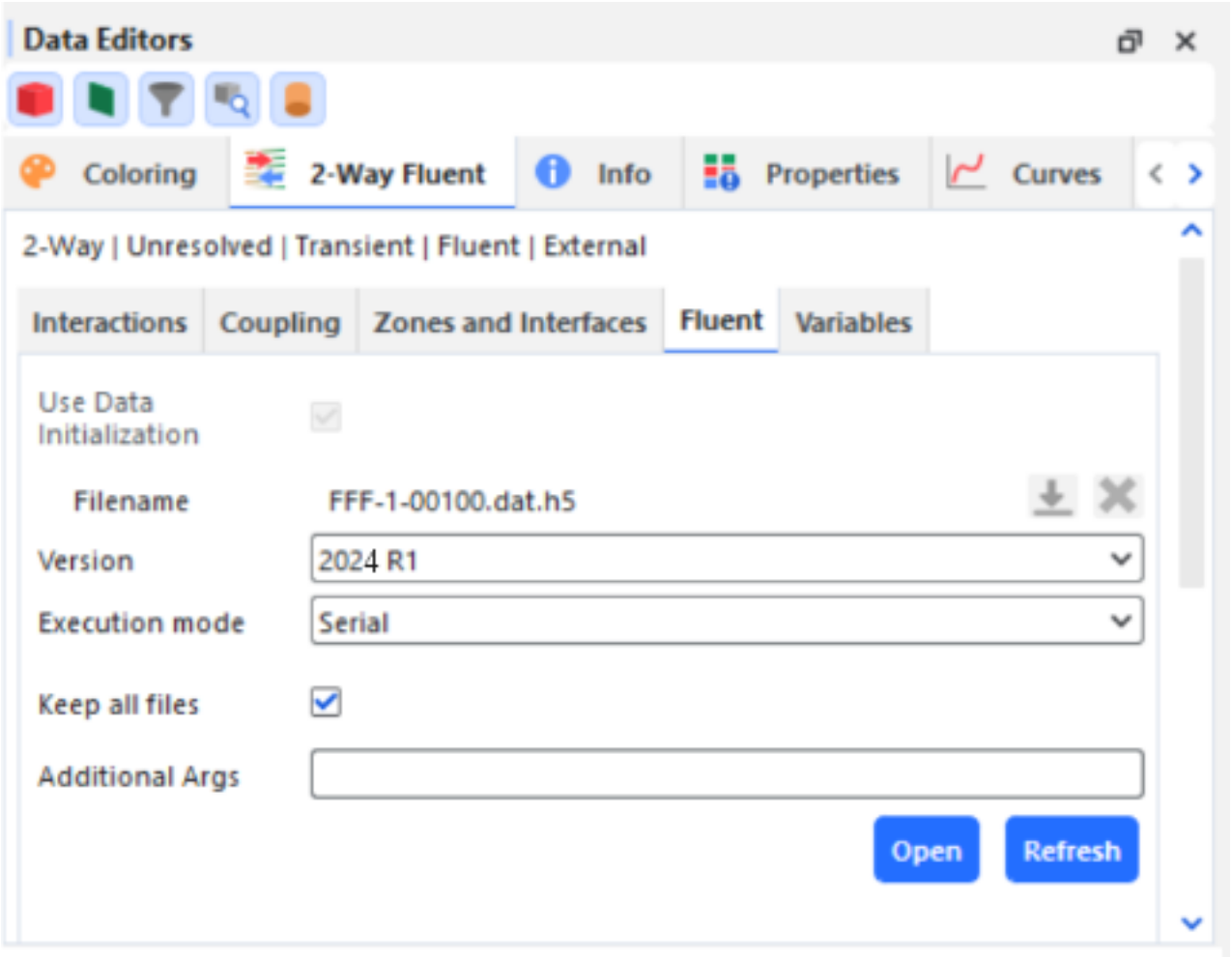
Figure 3.215: 2-way Coupled Rocky-Fluent simulation in Workbench initializing the fluid field using an initial fluid field data



The Fluent tab of the 2-Way Fluent editor will contain the Use Data Initialization checked and the Filename of the initial data that was imported into the coupled simulation.

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

Figure 3.216: Fluent tab in Rocky Data Editors with existing fluid data is used in a 2-way coupled simulation



Exchange (2-Way) Rocky Particle Data with Fluid Flow Data from Ansys Fluent

---

Tip:

This procedure covers only the final coupling portion of the entire 2-Way Fluent Coupling procedure, which applies to both multiphase (p. 1392) and single phase (p. 1395) coupling methods. To see the full procedure, including the setup steps that are required in Fluent, refer instead to Use the 2-Way Fluent Method to Process Fluent and Rocky Simulations (p. 386).

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

1. **Important:**

   Ensure that you have a Rocky-supported version of Ansys Fluent installed upon the same machine that contains your Rocky program installation. (See also System Requirements (p. 67).) Note: If you are not certain whether you have installed the **Ansys Fluent Coupling Support** component during your Rocky installation, first follow the Install Ansys Coupling Components (p. 792) procedure before beginning these steps.

2. From the **Data** panel, click **CFD Coupling** and then from the **Data Editors** panel, choose **Fluent** from the **Coupling Mode** list.

3. From the **Select Fluent CAS file** dialog, locate and select the CAS (CAS.H5 or CAS.GZ) file you previously generated in Fluent, and then click **Open**. Important: A mesh validation step will occur immediately after the CAS file import. This requires a valid Fluent license on the same machine upon which you are running the Rocky simulation. After successful import, a new **2-Way Fluent** item appears beneath **CFD Coupling** in the **Data** panel. In addition, if you chose to set up Moving Meshes in Fluent, one new Frame will appear under **Motion Frames** in the **Data** panel for each Cell Zone with motions you have defined. Important: If you have defined your Moving Meshes in Fluent using expressions, Rocky will not be able to create the associated Motion Frames for you automatically. Rather, you must manually create your own Motion Frames in Rocky to match the Fluent motions as closely as possible.

4. If the CAS (CAS.H5 or CAS.GZ) imported new Motion Frames (or you have created them yourself manually), assign each new Frame to the geometry component you want to share corresponding movements with Fluent. (See also Apply a Motion Frame to an Imported Geometry (p. 569).)

5. From the **Data** panel under **CFD Coupling**, select the new **2-Way Fluent** item and then from the **Data Editors** panel, do the following:

   a. From the **Interactions** tab, define the CFD laws you want per Particle set (p. 1393) by selecting (or multi-selecting) the set name(s) from the **Particle** list, and then defining the laws you want for the selected set(s). From this tab, you can also set the **Turbulent Dispersion** options for the entire coupled simulation. (See also About Using the 2-Way Fluent Method (p. 795).)

   b. From the **Coupling** tab, define the **Mapping Method** and **Sub-Stepping** options you want.

   c. From the **Zones and Interfaces** tab, do the following:

      - Select one or more options from the **Coupling Fluid Cell Zone** list.
      - (Optional) If provided, select the options you want from the **Mapping Cell Zone Interfaces** list.

   d. From the **Fluent** tab, do all of the following:

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

i. **Important:**

If your Fluent case is multiphase (p. 1392), from the **Rocky Phase** list, ensure that the phase representing particle flow is chosen.

ii. Choose what fluid flow data Rocky starts with by doing one of the following:

- To start your Rocky simulation without providing initial flow field data for the fluid, ensure the **Use Data Initialization** checkbox remains cleared. Doing so ensures that the initialization settings prescribed in Fluent will be used to generate the initial fluid flow.
- To start your Rocky simulation with initial fluid flow data, select **Use Data Initialization**, click the **Import File** button, and then select and open the DAT (DAT.H5 or DAT.GZ) file you saved in Step II. Important: If you chose to use initial fluid flow data in your simulation and your simulation is multiphase (p. 1392), you must have ensured that the secondary phase (particulate phase) had a volume fraction of zero set before you saved your DAT (DAT.H5 or DAT.GZ) file. This is important because later, after the simulation is initialized, the initial particulate volume fraction will be calculated in Rocky according to the particles enabled at the beginning of the simulation.

iii. From the **Version** list, select what Ansys version you want to use for fluid coupling. Tip:If you do not see the Ansys version you expected, see also Rocky does not list my Ansys version when I try to set up 2-Way Fluent Coupling (p. 1326).

iv. Choose how the fluid portion of the simulation is processed by doing one of the following:

- To have it processed on only one processor on your local machine, from the **Execution mode** list, select **Serial**.
- To have it processed on multiple processors on your local machine, from the **Execution mode** list, select **Local Parallel** and then from the **Solver Processes** field, enter the number of processors you want to dedicate to the CFD solver.
- To have it processed across several machines on your network, from the **Execution mode** list, select **Distributed Parallel** and then do one of the following:
  - Under **Hosts**, click the **Add** button for each server you want to add and then for each row, define the **Host name** and **Amount** information.
  - To import an external list of hosts and amounts that you created outside of Rocky, click the **Import File** button, and then from the **Import** dialog, locate and select the TXT file containing the information, and then click **Open**. The **Host** section is populated with the information from the TXT file. Tip:The format for the TXT file needs to include the **Host name** repeated on separate lines for each **Amount** you want devoted. To see an example, create a few sample rows as specified directly above, click **Export File** and then save and view the results as a TXT file.

v. Choose how many Fluent files to keep by doing one of the following:

- To keep each Fluent DAT (DAT.H5 or DAT.GZ) file (and in some cases, as with moving meshes, CAS files as well) for every Rocky output file that is saved, enable the **Keep all files** checkbox. Important: Unless you require all files for post-processing in Fluent,

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

keeping all files is not recommended for the 2-Way Fluent method. (See also About Using the 2-Way Fluent Method (p. 795).)

- To keep only the last (or last several) Fluent DAT (DAT.H5 or DAT.GZ) (and sometimes CAS) files saved, ensure the **Keep all files** checkbox is cleared, and then enter the number of last-saved files you want to keep in the **Files to keep** box. Tip:Keeping the last 2 (or more) files is recommended. Important: If you decide to save fewer than all of the files, be aware that you will not be able to post-process the unsaved files in Fluent. Also, be aware that the number of DAT (and sometimes CAS) files you save will be as far back in time as you can restart your simulation since fluid data will not be available for prior times. Tip:You can also choose which quantities and frequency you want to write to disk using Fluent's Autosave functionality.

e. From the **Variables** tab, review the **Additional Inputs** and **Additional Outputs** (if applicable) that Rocky and Fluent will exchange during the coupled simulation.

Tips:

- To verify or change the CAS (CAS.H5 or CAS.GZ) file selected, click **Open** (at the bottom of the **2-Way Fluent** | **Fluent** tab) and Rocky will open its own copy of the CAS (CAS.H5 or CAS.GZ) file in Fluent. From here, you can view or make changes to the Rocky copy of the file. Important: Because Rocky uses only its own copy of the imported CAS (CAS.H5 or CAS.GZ) file, it is critical that you only open the imported CAS (CAS.H5 or CAS.GZ) from Rocky's **Open** button, and that you save any changes to the default location Rocky indicates.

- If you make changes to the Rocky copy of the CAS (CAS.H5 or CAS.GZ) file in Fluent, click **Refresh** to have Rocky include the updates. Important: The **Refresh** button works only if you first opened the CAS (CAS.H5 or CAS.GZ) file from the Rocky **Open** button located at the bottom of the **2-Way Fluent** | **Fluent** tab.

- If you made changes to your original Fluent version of the CAS (CAS.H5 or CAS.GZ) file and did not use the Rocky **Open** button as specified above, you must set the **CFD Coupling** option to **No Coupling** to clear the values, and then restart this procedure again from the beginning, selecting the updated CAS (CAS.H5 or CAS.GZ) file in step 3.

6. Keeping in mind the **Time Step Size** value you set in Fluent, from the **Data** panel, click **Solver** and then from the **Data Editors** panel, select the **Time** tab and ensure that what you want is set for **Fluent Outputs Multiplier**. (See also About Solver Parameters (p. 464).) Tip:You can see how what you entered affects Rocky's Output Frequency in the disabled **Simulation** field.

See Also:

- About Using the 2-Way Fluent Method (p. 795)
- Use the 2-Way Fluent Method to Process Fluent and Rocky Simulations (p. 386)

USE THE 2-WAY FLUENT SEMI-RESOLVED METHOD

The 2-Way Fluent Semi-Resolved Method is a CFD Coupling option that is especially designed to be used with very large (when compared to the cell size of the Fluent mesh) particles that are interacting with a dynamic fluid whose cell mesh is many times smaller than the particle size. It is most useful

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

in cases where the resolution of the fluid flow around the particles is critical to capturing the behavior of your particles.

Note:

There are no special components that must be installed in Rocky for conducting a 2-Way coupled simulation with Fluent.

What would you like to do?

- Learn more About Using the 2-Way Fluent Semi-Resolved Method (p. 831)
- Exchange (2-Way) Rocky Particle Data with Semi-Resolved Fluid Flow Data from Ansys Fluent (p. 837), which covers only the final coupling portion of the entire 2-Way Fluent Coupling procedure.
- Use the 2-Way Fluent Semi-Resolved Method to Process a Fluent and Rocky Simulation (p. 392), which covers the full procedure, including the setup steps that are required in Fluent.

See Also:

- Set or Modify Fluid and/or Air Flow Properties (p. 757)
- Use the 1-Way LBM Method (p. 770)
- Use the 1-Way Constant Method (p. 763)
- Use the 1-Way Fluent Method (p. 775)
- Use the 2-Way Fluent Method (p. 795)
- Set Simulation Parameters (p. 414)

About Using the 2-Way Fluent Semi-Resolved Method

The 2-Way Fluent Semi-Resolved Method is a CFD Coupling option that is similar to the standard, 2-Way Fluent Method (p. 795) but rather than relying on standard correlations for fluid forces based on the centroid fluid velocity and velocity gradients, fluid forces are integrated at the particle's surface. This requires a very fine mesh around the particle in order to work properly.

This method is therefore designed to be used with very large (when compared to the cell size of the Fluent mesh) particles, that are interacting with a dynamic fluid whose cell mesh is many times smaller than the particle size. It is most useful in cases where the resolution of the fluid flow around the particles is critical to capturing the particle behavior.

CHOOSING THE CORRECT 2-WAY FLUENT METHOD

When determining which 2-Way Fluent method to use, consider the following questions:

1. Does your application require a very refined Fluent mesh in order to accurately simulate the fluid flow behavior?
2. Are you able to define your Fluent mesh without the use of polyhedral (including CutCell type) cells? (Can you use either tetrahedrons, prisms, pyramids, or hexahedrons?)

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

3. Does your coupled simulation require particles that are at least 10 times larger than the size of the fluid mesh cell?

4. Do you plan to have fewer than 1000 total particles in your coupled simulation?

5. Do you plan to use only Spheres, Shells (either rigid or flexible), or Custom Imported Polyhedrons (rigid only) for your particle shapes?

6. Are you able to get the results you need without the use of the CFD Coupling Particle Statistics (p. 873) module?

If the answers to all of the statements above are "yes", then the semi-resolved method might be the best method for your coupled simulation.

If you answered "no" to any of the questions above, the standard 2-Way Fluent Method (p. 795) might work better for your needs.

Use the information in the sections below to learn more about using the 2-Way Fluent Semi-Resolved Method.

SUPPORTED ABILITIES

Similar to the standard 2-Way Fluent Method, the 2-Way Semi-Resolved method supports both single and multiphase approaches. It is also similar in regards to the files and formats used.

Thermal support is enabled by activating the Thermal option in Rocky and the Energy Equation in Fluent. It is compatible with turbulence models from the κ-ε and κ-ω families and is only available for single-phase VOF simulations.

Refer to the related sections in the About Using the 2-Way Fluent Method (p. 795) topic for details.

UNSUPPORTED ABILITIES

Unlike the standard 2-Way Fluent Method, the 2-Way Fluent Semi-Resolved method does not currently support the following abilities:

- Post-processing (within Rocky) of fluid-related Properties (Fluid Density, Fluid Velocity, Fluid Viscosity, etc.). (See also About Properties (p. 933).)

- Assembly (p. 1389) particle shapes, Fibers (p. 1391) (both rigid and flexible), default Solid particle shapes, and flexible custom imported Polyhedron particle shapes. (See also Rocky Particle Shapes and Capabilities (p. 41).)

- The CFD Coupling Particle Statistics Module. (See also About the CFD Coupling Particle Statistics Module (p. 873).) Because the 2-Way Fluent Semi-Resolved method does not compute the forces individually, this module when enabled will not return the fluid forces.

- Turbulent fluid flows.

- Periodic Domain.

- .

ABOUT SETTING OUTPUT FREQUENCY IN 2-WAY FLUENT SEMI-RESOLVED COUPLING SIMULATIONS

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

Unlike the standard 2-Way Fluent method, the 2-Way Fluent Semi-Resolved method does not use the **Time Step Size** value that you set in Fluent to determine the initial Output Frequency of the Rocky files. This means that it is possible to have your Fluent outputs and your Rocky outputs happen at different times and frequencies, which might affect your ability to post-process the resulting data in both programs.

If you wish for your Rocky data and your Fluent data to better match in both programs, it is best to ensure that the **Time Step Size** in Fluent and the **Output Settings** | **Time Interval** value in Rocky are multiples.

ABOUT SHELL PARTICLES AND BUOYANCY

When Shell particles (either rigid or flexible) are used in 2-Way Fluent Semi-Resolved cases, Rocky calculates buoyancy forces using the gravity direction and the pressure gradient on both sides of the particle.

Note:

For the other particle types, the buoyancy is calculated together with the integration of pressure forces.

FLUENT FILES AND STORAGE

Similar to the standard 2-Way Fluent Method, you are given the option to **Keep all files** or determine how many of the last-saved DAT (and in some cases, as with moving meshes, CAS) **Files to keep**, which can help you manage your Fluent file storage.

Keeping all files is not recommended for the 2-Way Fluent Semi-Resolved method because the Fluent mesh files used in this coupling method are typically much larger than with other methods, which takes up more storage space per file.

Unless you require the full set of files for post-processing in Fluent, it is recommended that you save only the last two (or more) Fluent files.

Important:

If you decide to save fewer than all of the files, be aware that you will not be able to post-process the unsaved files in Fluent. Also, be aware that the number of DAT (and sometimes CAS) files you save will be as far back in time as you can restart your simulation since fluid data will not be available for prior times.

Tip:

You can also choose which quantities and frequency you want to write to disk using Fluent's Autosave functionality.

2-WAY FLUENT SEMI-RESOLVED SETTINGS IN ROCKY

Use the images and tables below to help you understand how to set up the Rocky portion of the 2-Way Fluent method.

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

Figure 3.217: CFD Coupling, 2-Way Fluent Semi-Resolved in the Data Editors panel

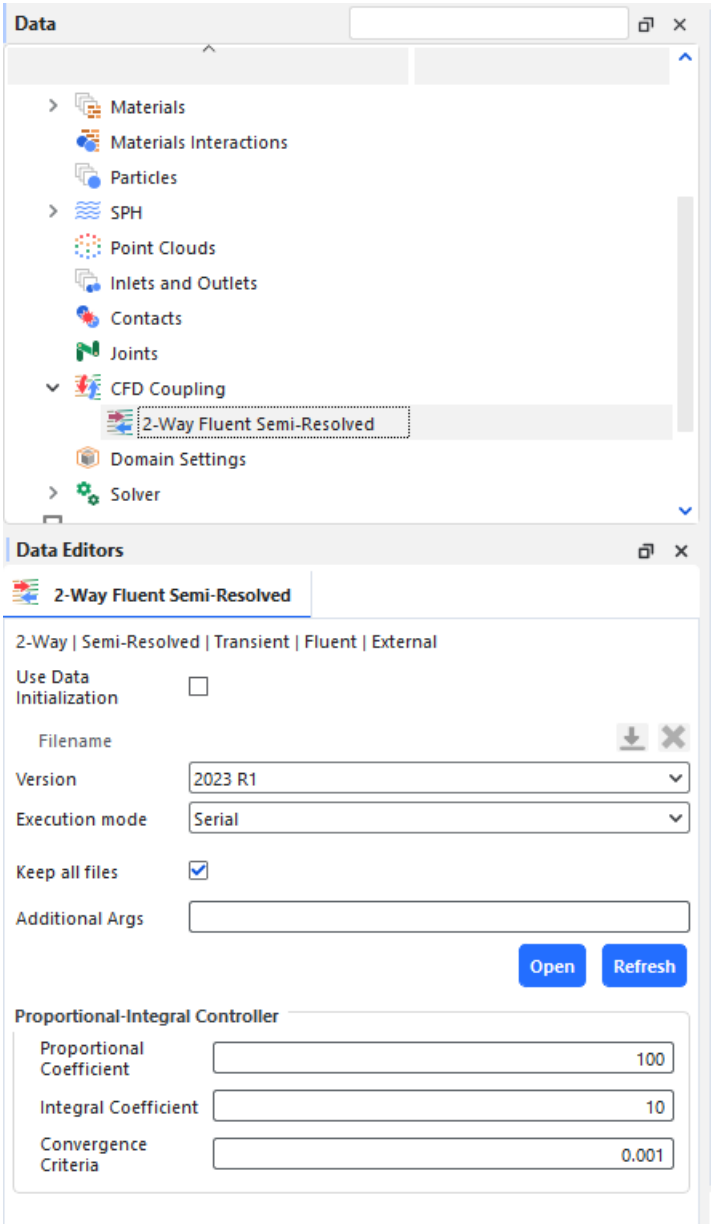


Table 1: 2-Way Fluent Semi-Resolved Settings

Setting	Description	Range
Use Data Initialization	When selected, enables you to begin your simulation with an initial flow field based upon the Fluent fluid phases. This initial solution is saved from Fluent into a DAT (DAT.H5 or DAT.GZ) file that is then used by Rocky during processing.	Turns on or off

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

Setting	Description	Range
Version	Enables you to select the Ansys Fluent software version you want to use for processing. **Tip:** If you do not see the Ansys version you expected, see also Rocky does not list my Ansys version when I try to set up 2-Way Fluent Coupling (p. 1326).	List automatically generated based upon the software versions detected.
Execution mode	Enables you to select how your computer and/or network will process the fluid portion of the coupled simulation. Note: Regardless of what is selected here, the Rocky portion of the coupled simulation will still be processed according to the **Processing Unit** and related information that you specified on **Solver General Settings** tab (see also About Starting a Simulation (p. 908). Specifically: • **Serial** allows a single Fluent process on your local computer to calculate the fluid portion of the coupled simulation. • **Local Parallel** allows you to select multiple Fluent processes on your local computer from which to calculate the fluid portion of the coupled simulation. • Distributed Parallel ** allows you to spread the fluid processing portion of the coupled simulation across multiple separate machines on your network.	Serial; Local Parallel; Distributed Parallel
Solver Processes	When **Local Parallel** is selected for **Execution mode**, this enables you to determine the number of computer processors (threads) you want dedicated to the CFD solver. **Note:** You select the number of processors used by Rocky when you start processing the simulation. (See also About Starting a Simulation (p. 908)). **Note:** This text field does not support parametric variables. (See also I cannot enter an input variable or mathematical function into a text field (p. 1334)).	Positive values

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

Setting	Description	Range
Host Name	When **Distributed Parallel** is selected for **Execution mode**, this is the name of the remote machine on your network. Tip:Contact your IT administrator to find out the host names of the fluid nodes on your network.	99 character limit
Amount	When **Distributed Parallel** is selected for **Execution mode**, for each separate **Host Name** listed, this is the number of computer processors you want dedicated to the CFD solver.	Positive values
Keep all files	When selected, all Fluent .dat files (and in some cases, as with moving meshes, .cas files as well) will be saved. When cleared, this enables you to select how many .dat (and sometimes .cas) **Files to keep**. Keeping fewer than all .dat (and sometimes .cas) files can help you save storage space when processing your simulation. However, doing so can also limit your fluid post-processing abilities, as well as your ability to restart the simulation from an earlier time due to the reduced amount of past fluid data that is available when you save fewer .dat (and sometimes .cas) files.	Turns on or off
Files to keep	When **Keep all files** is cleared, this determines how many Fluent .dat files (and in some cases, as with moving meshes, .cas files as well) are saved. For example, if set to 1, only the last .dat (and sometimes .cas) files will be kept. If set to 2, only the last 2 .dat (and sometimes .cas) files will be kept, and so on. This setting can be useful in cases where you need only to keep files past steady state, for example, or when you do not require the Fluent files for post-processing. **Important:** If you decide to save fewer than all of the files, the number of .dat (and sometimes .cas) files you save will be as far back in time as you can restart your simulation since fluid data will not be available for prior times. **Note:** This text field does not support parametric variables. (See also I cannot enter an input variable or mathematical function into a text field (p. 1334)).	Whole values greater than or equal to 1
Additional Args	This displays the set of arguments that Rocky passed "as-is" from the Fluent command line when the coupled simulation began. For advanced users, this enables more control of the Fluent portion of the processing. For more information about these arguments, see your Ansys Fluent documentation.	Positive values

What would you like to do?

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

- Exchange (2-Way) Rocky Particle Data with Semi-Resolved Fluid Flow Data from Ansys Fluent (p. 837), which covers only the final coupling portion of the entire 2-Way Fluent Coupling procedure.
- Use the 2-Way Fluent Semi-Resolved Method to Process a Fluent and Rocky Simulation (p. 392), which covers the full procedure, including the setup steps that are required in Fluent.

See Also

- Set or Modify Fluid and/or Air Flow Properties (p. 757)

Exchange (2-Way) Rocky Particle Data with Semi-Resolved Fluid Flow Data from Ansys Fluent

---

Tip:

This procedure covers only the final coupling portion of the entire 2-Way Fluent Semi-Resolved Coupling procedure, which applies to both multiphase (p. 1392) and single phase (p. 1395) coupling methods. To see the full procedure, including the setup steps that are required in Fluent, refer instead to Use the 2-Way Fluent Semi-Resolved Method to Process a Fluent and Rocky Simulation (p. 392).

---

1. **Important:**

   Ensure that you have a Rocky-supported version of Ansys Fluent installed upon the same machine that contains your Rocky program installation. (See also System Requirements (p. 67).) Note: If you are not certain whether you have installed the **Ansys Fluent Coupling Support** component during your Rocky installation, first follow the Install Ansys Coupling Components (p. 792) procedure before beginning these steps.

2. From the **Data** panel, click **CFD Coupling** and then from the **Data Editors** panel, choose **Fluent Semi-Resolved** from the **Coupling Mode** list.
3. From the **Select Fluent CAS file** dialog, locate and select the CAS (CAS.H5 or CAS.GZ) file you previously generated in Fluent, and then click **Open**. Important: A mesh validation step will occur immediately after the CAS file import. This requires a valid Fluent license on the same machine upon which you are running the Rocky simulation. After successful import, a new **2-Way Fluent Semi-Resolved** item appears beneath **CFD Coupling** in the **Data** panel. In addition, if you chose to set up Moving Meshes in Fluent, one new Frame will appear under **Motion Frames** in the **Data** panel for each Cell Zone with motions you have defined. Important: If you have defined your Moving Meshes in Fluent using expressions, Rocky will not be able to create the associated Motion Frames for you automatically. Rather, you must manually create your own Motion Frames in Rocky to match the Fluent motions as closely as possible.
4. If the CAS (CAS.H5 or CAS.GZ) file imported new Motion Frames, assign each newly imported Frame to the geometry component you want to share corresponding movements with Fluent. (See also Apply a Motion Frame to an Imported Geometry (p. 569).)
5. From the **Data** panel under **CFD Coupling**, select the new **2-Way Fluent Semi-Resolved** item and then from the **Data Editors** panel, do all of the following:

   a. Choose what fluid flow data Rocky starts with by doing one of the following:

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

- To start your Rocky simulation without providing initial fluid flow data, ensure the **Use Data Initialization** checkbox remains cleared. Doing so ensures that the initialization settings prescribed in Fluent will be used to generate the initial fluid flow.

- To start your Rocky simulation with initial fluid flow data, select **Use Data Initialization**, click the **Import File** button, and then select and open the DAT (DAT.H5 or DAT.GZ) file you want.

b. From the **Version** list, select what Ansys version you want to use for fluid coupling. Tip:If you do not see the Ansys version you expected, see also Rocky does not list my Ansys version when I try to set up 2-Way Fluent Coupling (p. 1326).

c. Choose how the fluid portion of the simulation is processed by doing one of the following:

- To have it processed on only one processor on your local machine, from the **Execution mode** list, select **Serial**.

- To have it processed on multiple processors on your local machine, from the **Execution mode** list, select **Local Parallel** and then from the **Solver Processes** field, enter the number of processors you want to dedicate to the CFD solver.

- To have it processed across several machines on your network, from the **Execution mode** list, select **Distributed Parallel** and then do one of the following:

  – Under **Hosts**, click the **Add** button for each server you want to add and then for each row, define the **Host name** and **Amount** information.

  – To import an external list of hosts and amounts that you created outside of Rocky, click the **Import File** button, and then from the **Import** dialog, locate and select the TXT file containing the information, and then click **Open**. The **Host** section is populated with the information from the TXT file. Tip:The format for the TXT file needs to include the **Host name** repeated on separate lines for each **Amount** you want devoted. To see an example, create a few sample rows as specified directly above, click **Export File** and then save and view the results as a TXT file.

d. Choose how many Fluent files to keep by doing one of the following:

- To keep each Fluent DAT (DAT.H5 or DAT.GZ) file (and in some cases, as with moving meshes, CAS files as well), enable the **Keep all files** checkbox. Important: Unless you require all files for post-processing in Fluent, keeping all files is not recommended for the 2-Way Fluent Semi-Resolved method. (See also About Using the 2-Way Fluent Semi-Resolved Method (p. 831).)

- To keep only the last (or last several) Fluent DAT (DAT.H5 or DAT.GZ) (and sometimes CAS) files saved, ensure the **Keep all files** checkbox is cleared, and then enter the number of last-saved files you want to keep in the **Files to keep** box. Tip:Keeping the last 2 (or more) files is recommended. Important: If you decide to save fewer than all of the files, be aware that you will not be able to post-process the unsaved files in Fluent. Also, be aware that the number of DAT (and sometimes CAS) files you save will be as far back in time as you can restart your simulation since fluid data will not be available for prior times. Tip:You can also choose which quantities and frequency you want to write to disk using Fluent's Autosave functionality.

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

Tips:

- To verify or change the CAS (CAS.H5 or CAS.GZ) file selected, click **Open** and Rocky will open its own copy of the CAS (CAS.H5 or CAS.GZ) file in Fluent. From here, you can view or make changes to the Rocky copy of the file. Important: Because Rocky uses only its own copy of the imported CAS (CAS.H5 or CAS.GZ) file, it is critical that you only open the imported CAS (CAS.H5 or CAS.GZ) from Rocky's **Open** button, and that you save any changes to the default location Rocky indicates.
- If you make changes to the Rocky copy of the CAS (CAS.H5 or CAS.GZ) file in Fluent, click **Refresh** to have Rocky include the updates. Important: The **Refresh** button works only if you first opened the CAS (CAS.H5 or CAS.GZ) file from the Rocky **Open** button.
- If you made changes to your original Fluent version of the CAS (CAS.H5 or CAS.GZ) file and did not use the Rocky **Open** button as specified above, you must set the **CFD Coupling** option to **No Coupling** to clear the values, and then restart this procedure again from the beginning, selecting the updated CAS (CAS.H5 or CAS.GZ) file in step 3.

See Also:

- About Using the 2-Way Fluent Semi-Resolved Method (p. 831)
- Use the 2-Way Fluent Semi-Resolved Method to Process a Fluent and Rocky Simulation (p. 392)

3.2.17. Define External Coupling Options

If your Rocky project was opened through Workbench, an **External Coupling** component automatically appears in the Rocky **Data** panel. For Rocky projects coupled with Ansys Mechanical through Workbench, this component enables you to define the kind of data gets shared between the programs, such as the geometry loads.

For Rocky projects that are coupled with other Ansys products (not including Mechanical) through Workbench, this component can be ignored.

What would you like to do?

- Learn more About Defining External Coupling Options (p. 839)
- Define Options for External Coupling (p. 842)

See Also:

- Understand Rocky and Ansys Integration (p. 143)

ABOUT DEFINING EXTERNAL COUPLING OPTIONS

If you have an **External Coupling** component in your Rocky **Data** panel, this indicates that your Rocky project has been opened through Workbench. Although it will appear for any type of coupled project

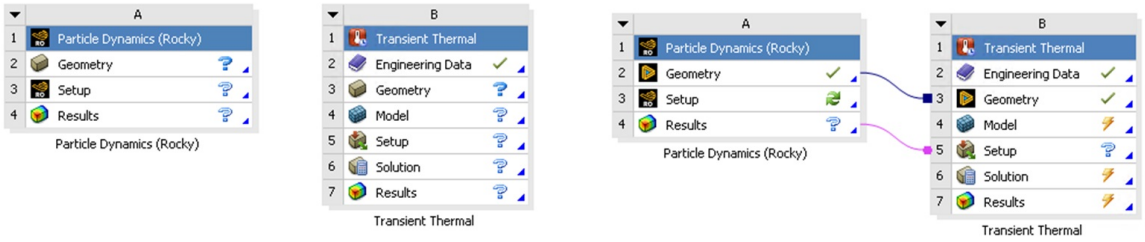
Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

that you open through Workbench, it is designed to work only with Rocky projects that are coupled specifically with Ansys Mechanical.

Tip:

As a best practice, it is important to open workbench, insert the Rocky and Mechanical components (left figure below showing unattached Rocky and Transient Thermal), then manually open an save Rocky's project, and only then manually link Rocky to Mechanical (right figure below showing Rocky and Transient Thermal attached).

Figure 3.218: Workbench best practices for Rocky-Mechanical coupling.



For projects coupled with Mechanical, defining the options for this component enables you to determine what kind of particle data to share, as described below.

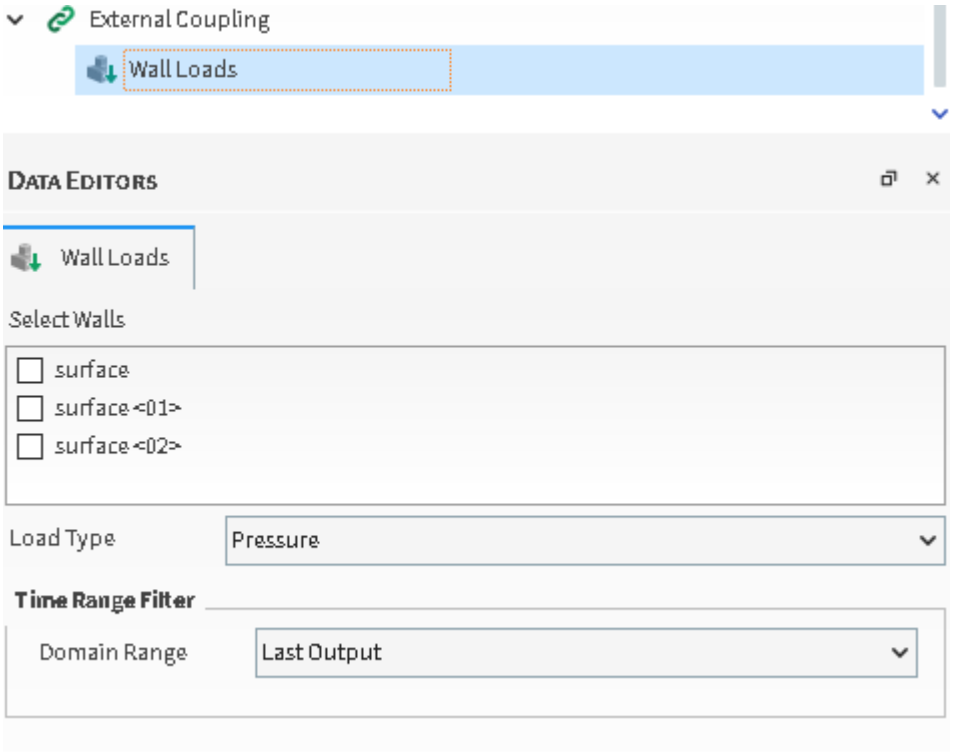
For all other coupled projects, including those with only SpaceClaim or with only 1-Way Fluent, this **External Coupling** component can be ignored.

About the Geometry Loads Component

When you define options for the **Geometry Loads** component, you choose the geometry component(s) for which you want to share particle load data with the coupled Ansys Mechanical program, and the simulation time frame during which you want data (Figure 1).

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

Figure 3.219: External Coupling entity expanded to show the Geometry Loads component



By limiting the data you share to only what you require for your analyses, you reduce the amount of time the coupled simulation will take to process.

GEOMETRY LOAD PARAMETERS

Use the image above and table below to help you understand how to define options for Geometry Loads.

Table 1: Geometry Loads Parameter Definitions

Setting	Description	Range
Select Geometries	Enables you to choose which geometry data will be shared during the coupled simulation.	Turns on or off
Domain Range	Defines what time steps of the simulation are included in the data to be shared. The options are as follows: • **Application Time Filter**: Use the range settings on the Time Filter dialog to limit the time steps used. (For more information, see About the Time Toolbar.) • **All**: Use all available time steps in the simulation. • **Last Output**: Use only the very last (most recent) time step that was calculated. • **Time Range**: Limit the time step values by a specific range that you choose. (Uses the **Initial** and **Final** values you set.)	Application Time Filter; All; Last Output; Time Range; Specify Time; After Time; Time Range Relative to Simulation End

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

Setting	Description	Range
	• **Specific Time**: Choose only a single specific time step. (Uses the **At Time** value you set.) • **After Time**: Use all time steps available after a specific time period of your choosing. (Uses the **Initial** value you set.) • **Time Range Relative to Simulation End**: Specify how long before the very end of the simulation to include time steps. (Uses the **Range from end** value you set.)	
Initial	When **Time Range** or **After Time** is chosen for **Domain Range**, this is the starting time to begin the time step selection.	Any value between 0 and the final simulation time.
Final	When **Time Range** is chosen for **Domain Range**, this is the ending time to stop the time step selection.	Any value between the **Initial** time and the final simulation time.
At Time	When **Specific Time** is chosen for **Domain Range**, this is the exact moment in which the time step will be selected.	Any value between 0 and the final simulation time.
Range from end	When **Time Range Relative to Simulation End** is chosen for **Domain Range**, this is the period of time before the final simulation time in which time steps will be included in the selection. For example, when you want to include only the last X seconds of the simulation.	Any value between 0 and the final simulation time.

What would you like to do?

- Define Options for External Coupling (p. 842)

See Also:

- About Rocky and Ansys Workbench Integration (p. 144)

DEFINE OPTIONS FOR EXTERNAL COUPLING

Note:

Even though the **External Coupling** component appears for any type of Rocky project that is opened through Ansys Workbench, it only functions with projects that are coupled specifically with Ansys Mechanical.

---

1. Ensure that your Mechanical-coupled Rocky project has been opened through Workbench. (See also About Rocky and Ansys Workbench Integration (p. 144).)
2. From the **Data** panel, under **External Coupling**, select **Geometry Loads**.

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

3. From the **Data Editors** panel, under **Select Geometries**, enable the checkbox for each geometry component for which you want to share data with the coupled Mechanical program.

4. Under **Time Range Filter**, choose the **Domain Range** and related options that define the time range during which you want to share data for the geometry component(s) you selected.

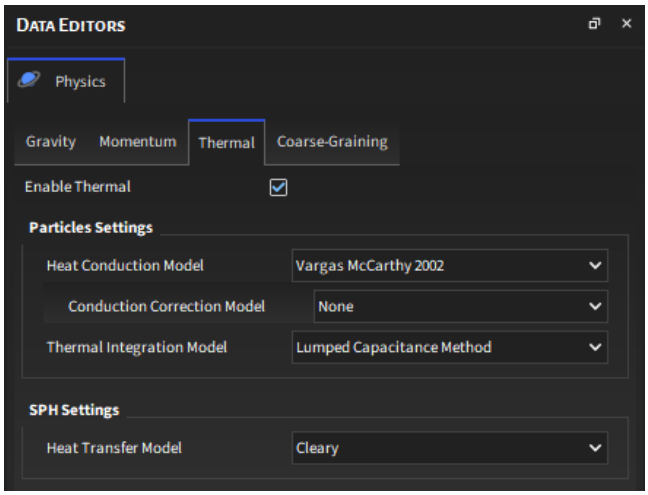
See Also:

- About Defining External Coupling Options (p. 839)
- About Rocky and Ansys Workbench Integration (p. 144)

ABOUT THE HEAT TRANSFER COEFFICIENT (HTC) COMPONENT

When you wish to automatically transfer Rocky's HTC calculations to Mechanical, when connecting the Rocky system to a Steady-State or a Transient Thermal using Workbench, it is necessary to activate the Thermal calculations in Rocky's Physics (Physics thermal tab when HTC transfer between Rocky and Mechanical via Workbench is used. (p. 843))

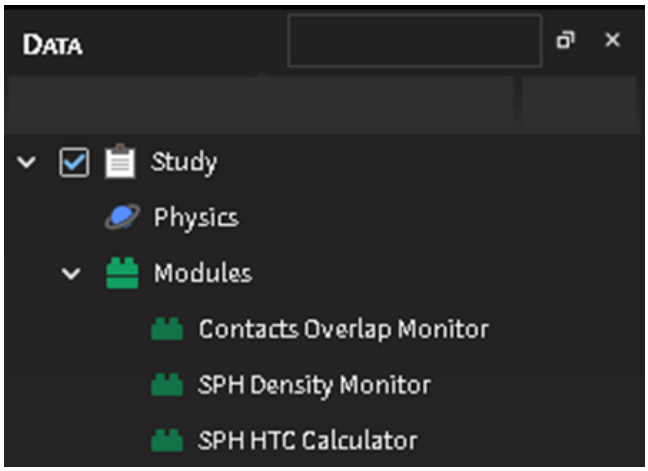
Figure 3.220: Physics thermal tab when HTC transfer between Rocky and Mechanical via Workbench is used.



The module called **SPH HTC Calculator** should also be enabled:

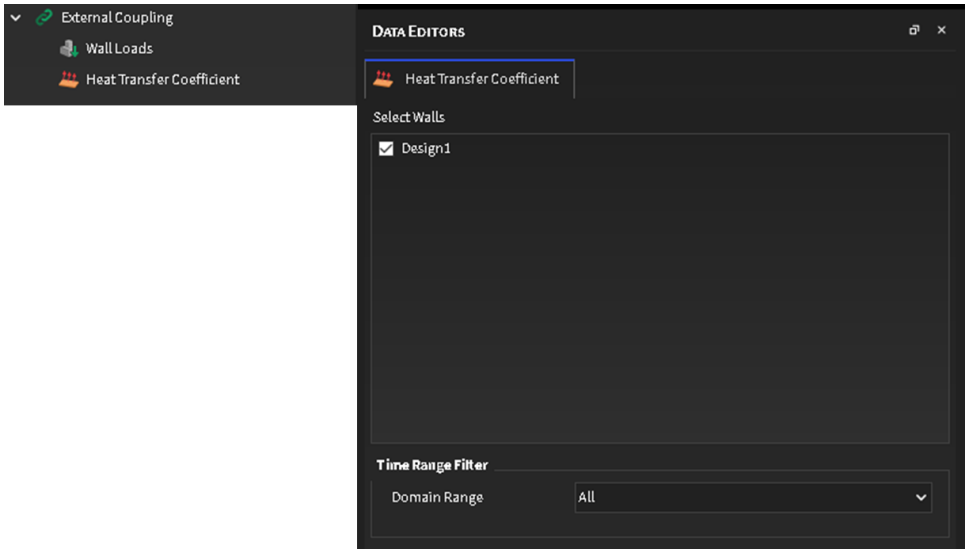
Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

Figure 3.221: SPH HTC Calculator module enabled.



This enables another component in the **External Coupling**, the **Heat Transfer Coefficient**, which lets you assign the **walls** for which the HTC shall occur and the **Time Range Filter** for the transfer to happen:

Figure 3.222: Physics thermal tab when HTC transfer between Rocky and Mechanical via Workbench is used.



Heat Transfer Coefficient Parameters

Use the image above and table below to help you understand how to define options for the HTC transfer.

Table 1: Heat Transfer Coefficient Parameter Definitions

Setting	Description	Range
Select Walls	Enables you to choose which geometry data will be shared during the coupled simulation.	Turns on or off

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

Setting	Description	Range
Time Range Filter	Defines what time steps of the simulation are included in the data to be shared. The options are as follows: • **Application Time Filter**: Use the range settings on the Time Filter dialog to limit the time steps used. (For more information, see About the Time Toolbar.) • **All**: Use all available time steps in the simulation. • **Last Output**: Use only the very last (most recent) time step that was calculated. • **Time Range**: Limit the time step values by a specific range that you choose. (Uses the **Initial** and **Final** values you set.) • **Specific Time**: Choose only a single specific time step. (Uses the **At Time** value you set.) • **After Time**: Use all time steps available after a specific time period of your choosing. (Uses the **Initial** value you set.) • **Time Range Relative to Simulation End**: Specify how long before the very end of the simulation to include time steps. (Uses the **Range from end** value you set.)	Application Time Filter; All; Last Output; Time Range; Specify Time; After Time; Time Range Relative to Simulation End
Initial	When **Time Range** or **After Time** is chosen for **Domain Range**, this is the starting time to begin the time step selection.	Any value between 0 and the final simulation time.
Final	When **Time Range** is chosen for **Domain Range**, this is the ending time to stop the time step selection.	Any value between the **Initial** time and the final simulation time.
At Time	When **Specific Time** is chosen for **Domain Range**, this is the exact moment in which the time step will be selected.	Any value between 0 and the final simulation time.
Range from end	When **Time Range Relative to Simulation End** is chosen for **Domain Range**, this is the period of time before the final simulation time in which time steps will be included in the selection. For example, when you want to include only the last X seconds of the simulation.	Any value between 0 and the final simulation time.
Thermal Settings	**Reference Temperature**: defines the reference temperature for the HTC Calculation	Any value.

What would you like to do?

- Define Options for External Coupling (p. 842)

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

See Also:

- About Rocky and Ansys Workbench Integration (p. 144)

3.2.18. Duplicate a Data Panel Item

When you duplicate an item in the **Data** panel, you make an exact copy that you can then use as a starting place for making a similar but slightly different item, which can be quicker than starting from a brand new item.

You may duplicate many items in the **Data** panel including individual geometry components, individual material components, individual Particle sets, particle inputs, individual User Processes, and Motion Frames.

TO DUPLICATE A DATA PANEL ITEM:

1. Ensure that the item you want to duplicate is already created and set up the way you want.
2. From the **Data** panel, right-click the item you want to copy, and then choose **Duplicate**. A copy of the item appears in the same list as the original item you chose, the name of which is appended with a new number.

See Also:

- Set Simulation Parameters (p. 414)

3.2.19. Remove Multiple Items from the Data Panel

If you have the need to remove many separate items under the same Data panel category, you may remove all those items at one time with multi-select. These include many items in the **Data** panel such as individual geometry components, individual material components, individual Particle sets, particle inputs, and individual User Processes.

TO REMOVE MULTIPLE ITEMS FROM THE DATA PANEL:

1. Under the **Data** panel category of your choice, use your mouse to multi-select all the individual components that you want to remove. Note: The multiple items must all belong to the same Data panel category.
2. Right-click the group of items you've selected, and then choose the **Remove** option. The items you selected will be removed from the list.

See Also:

- Set Simulation Parameters (p. 414)

3.2.20. Set Up Parameters When Restarting a Simulation

Once you save a copy of your partially processed simulation (see also Save a Copy of a Partially Processed Simulation for Restart Purposes (p. 855)), you are able to change or delete existing boundaries and/or Particle sets as well as add new ones.

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

However, to preserve the integrity of the original simulation, Rocky limits the setup parameters that you can change for existing geometries and Particle sets to only a select few.

What would you like to do?

- Learn more About Changing Geometries and/or Particles When Restarting a Simulation (p. 847)
- Change Geometries and/or Particles When Restarting a Simulation (p. 851)

See Also:

- View Frequently Used Tasks (p. 371)

ABOUT CHANGING GEOMETRIES AND/OR PARTICLES WHEN RESTARTING A SIMULATION

After saving a copy of a partially processed simulation for restart purposes (see also Save a Copy of a Partially Processed Simulation for Restart Purposes (p. 855)), you can add additional geometries and Particle sets (p. 1393) but will be unable to change some settings for existing geometries, inlets, and particles due to the way they affect the calculations that are already in process. Most other settings will be unaffected in the copy and can be modified as usual (Table 1).

Note:

If you have saved for restart purposes a copy of a partially processed simulation that had both breakage enabled and broken fragments (p. 1393) within the simulation at the time you saved your copy, the fragments will not be present in the copy you want to restart due to a limitation between the features. However if you choose to restart processing the simulation copy, breakage should work in the newly processed output times. (See also Enable Particle Breakage Calculations (p. 397).)

See the sections below for further detail on what settings and parameters are editable within a copy of a partially processed simulation.

Changing Particle Inputs When Restarting a Simulation

Since any Volume Fill Input will inject all of its particles immediately after the simulation begins, and a simulation must have particles in it to be saved for restart, the particles injected from a Volume Fill input will still appear in the copy. However, because the injection has already completed, the settings for any existing Volume Fill inputs will be disabled. You are able to add new Volume Fill inputs or duplicate any existing Volume Fill inputs as needed.

Any Continuous Injection or Custom Inputs (see also About Adding and Editing Particle Inputs (p. 708)) you had in the original simulation that were active at the time the copy was saved will still appear in the copy. And while these copied inputs and the Particle sets (p. 1393) assigned to them cannot be removed from the copy, there are ways you can disable them if you no longer want to use them in the simulation moving forward.

Tips:

- If you do not want an individual Particle set within a copied Continuous Injection input to be released in the simulation copy, you can set the **Mass Flow Rate** of that Particle set to zero.

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

- If you have set all of the Particle sets within an individual Continuous Injection input to have zero as their Mass Flow Rates, you may choose to include a second Particle Input using the same Entry Point as the first, as long as all of the Mass Flow Rates in the first Input are zero, and all the Mass Flow Rates in the second Input are greater than zero.

Changing Motion Frames When Restarting a Simulation

Any Motion Frames (see also About Creating and Applying Motion Frames (p. 533)) you had in the original simulation that were active at the time the copy was saved will still appear in the copy and will retain their original geometry assignments. However, even though the geometries using the frames can be removed, the copied frames themselves cannot be removed from the copy, and will also only allow changes to the **Stop Time** field for the existing motion (as long as the copied motion hasn't completed yet, meaning the **Stop Time** value is positive); new motions can be added to the existing frame, however, by clicking the **Add Motion** button. In addition, only new motion frames can be assigned to new geometries in the copy; newly added geometries cannot be assigned frames copied from the original simulation, and geometries with motion frames copied from the original simulation cannot be assigned newly added motion frames.

Tips:

- If you do not want a copied motion to occur in the copy, or want to define a new kind of motion for the copied geometry that is assigned the copied frame, you can set the **Stop Time** of the copied motion in the copied frame to zero (0), and then create a new motion (click the **Add motion** button) within the existing frame if desired. **Note**: Setting the Motion Frame **Stop Time** to zero (0) disables the field from being changed further. If you need to edit the **Stop Time**, press the **Undo** button (see also Undo or Redo a Single Action at Once (p. 108)) to re-enable the field.

- If you want to change the motion assignment of an existing geometry, import a new copy of the geometry, create a new motion frame, and then assign the new motion frame to the new geometry. You can then delete the old geometry if you choose.

- If you want to assign a copied motion to a newly added geometry, make a copy of the old motion frame and then assign that to the new geometry.

Editable Settings When Restarting a Simulation

The following table attempts to document what Rocky settings are able to be changed in a copy of a partially processed simulation.

Table 1: Editable Settings When Restarting a Simulation

DATA PANEL SECTION	EDITABLE SETTINGS	CONDITIONS/NOTES	SEE ALSO
STUDY			
Study	All settings are editable.	None.	About Study Parameters (p. 415)
PHYSICS			
Physics	All settings are editable.	None.	About Physics Parameters (p. 417)
GEOMETRIES			

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

DATA PANEL SECTION	EDITABLE SETTINGS	CONDITIONS/NOTES	SEE ALSO
Feed Conveyor; Imported Geometry	Just the **Thermal Boundary Type**.	For existing feed conveyors and imported geometries.	About Feed Conveyor Parameters ;
Feed Conveyor	Just the **Belt Profile** settings (excluding the number of rolls) and the **Belt Motion** settings.	For existing feed conveyors that have released particles.	About Feed Conveyor Parameters
Feed Conveyor; Inlet	All settings are editable.	For feed conveyors or inlets that have not yet released particles.	About Feed Conveyor Parameters; About Inlet Parameters
Receiving Conveyor	All settings are editable.	None.	About Receiving Conveyor Parameters
Imported Geometries	• Material • Disable Time • Mass settings • Wear settings (excluding disabling them if they were enabled in the original file).	Only these settings can be edited on already imported geometries. However, for newly imported geometries all settings are editable.	
MOTION FRAMES			
Motion Frames	Just the **Stop Time**.	For any existing motion of an existing Motion Frame that is in process or has yet to happen.	About Creating and Applying Motion Frames (p. 533)
MATERIALS			
Materials	All settings are editable.	None.	About Modifying Material Compositions (p. 591)
Materials Interactions	All settings are editable.	Editing these settings will affect the previously released particles what may lead to unrealistic behavior. Please,	About Modifying Materials Interactions and Adhesion Values (p. 597)

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

DATA PANEL SECTION	EDITABLE SETTINGS	CONDITIONS/NOTES	SEE ALSO
		evaluate the results carefully.	
SPH			
SPH	• Turbulence Type; • Viscosity Type; • Surface Tension Type (and sub parameters); • Boundary Type; • Positions Correction Type.	None.	
SPH > Eulerian Solution	Enable **Eulerian Solution**.	None.	
Fluid Inlet	Just the **Stop Time**.	None.	
Outlet	All settings are editable.	None.	
PARTICLES			
Particles	Just the **Rolling Resistance**; and the **Enable Rotations**.	Only these settings can be edited on existing particles. However, for newly particles all settings are editable.	About Adding and Editing Particle Sets (p. 602)
PARTICLE INPUTS			
Continuous Injection	Just the **Stop Time**.	For existing inlets and feed conveyors that have released particles, as long as the **Stop Time** value is greater than the original simulation time represented by the current output time.	About Adding and Editing Particle Inputs (p. 708)
CFD COUPLING			
CFD Coupling	All settings are editable.	None.	Set or Modify Fluid and/or Air Flow Properties (p. 757)

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

DATA PANEL SECTION	EDITABLE SETTINGS	CONDITIONS/NOTES	SEE ALSO
DOMAIN SETTINGS			
Domain Settings	All settings are editable.	None.	About Domain Settings Parameters (p. 452)
SOLVER			
Solver	All settings are editable.	None.	About Solver Parameters (p. 464)
PARTICLES CALCULATIONS			
Particles Calculations	All settings are editable.	None.	About Particles Calculations (p. 1242)
USER PROCESSES			
User Processes	All settings are editable.	None.	Filter Views and Data with User Processes (p. 1174)
COLOR SCALES			
Color Scales	All settings are editable.	None.	About Color Scales (p. 1086)

Note:

If you need to modify any of the above settings, saving a copy of the processed simulation might not work for you. Instead, return to your original simulation and reset it before changing any of the above parameters.

What would you like to do?

- Change Geometries and/or Particles When Restarting a Simulation (p. 851)

See Also:

- Change Set Up Parameters When Restarting a Simulation (p. 846)

CHANGE GEOMETRIES AND/OR PARTICLES WHEN RESTARTING A SIMULATION

1. Process and save part of a processed simulation. (See also Save a Copy of a Partially Processed Simulation for Restart Purposes (p. 855).) The copy will open with the **Time** toolbar slider reset to zero (0). However, the simulation will be processed to the last output time that you selected in the original simulation.

2. Using the lists of what you can and cannot change for guidance (see also About Changing Geometries and/or Particles in a Partially Processed Simulation (p. 847)), change and/or delete existing boundaries, add new boundaries, and/or add new Particle sets as you want.

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

3. Process and analyze the simulation as you normally would.

See Also:

- View Frequently Used Tasks (p. 371)

3.3. Saving a Simulation Project

When you save a simulation project, you are saving into a .rocky file the various parameters you have specified in the Rocky user interface. When you choose to process a simulation, you will be asked to save the simulation project before processing. As you process, the simulation output files are saved automatically.

There are several different ways you can choose to save, copy, and archive your Rocky project, depending upon your goals. See sections below for details.

3.3.1. Save a New Project

You can save the full project, including any results that have been generated, as a new project by specifying a project name and location for the .rocky and related project files. (See also Save a New Simulation Project (p. 853).)

Once the project has been saved, you can continue to Save Over the Current Simulation Project (p. 857) as needed.

3.3.2. Save a Copy of a Project

If you want to save a copy of all or part of the Rocky project for building upon later, you have three options for doing so:

- Save an Exact Copy of the Simulation Project; Including the Simulation Results (p. 853)
- Save a Copy of Only the Simulation Settings; Not the Simulation Results (p. 854)
- Save a Copy of a Partially Processed Simulation for Restart Purposes (p. 855)

3.3.3. Save an Archive of the Project

Archiving a project is similar to saving a copy of the simulation, but packages it in a format easier sharing with support. (See also Archive a Simulation Project (p. 1294).)

3.3.4. Save an Image of a Window

You can also choose to save an image of what is currently being shown in the Workspace. Save an image when you need only a static snapshot of one moment during the simulation. (See also Save an Image of the Rocky Window (p. 858).)

What would you like to do?

- Save a New Simulation Project (p. 853)

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

- Save an Exact Copy of the Simulation Project; Including the Simulation Results (p. 853)
- Save a Copy of Only the Simulation Settings; Not the Simulation Results (p. 854)
- Save a Copy of a Partially Processed Simulation for Restart Purposes (p. 855)
- Save Over the Current Simulation Project (p. 857)
- Archive a Simulation Project (p. 1294)
- Save an Image of the Rocky Window (p. 858)

See Also:

- Setting Up a Simulation (p. 411)

3.3.5. Save a New Simulation Project

1. From the **File** menu, click **Save**.
2. In the **Save in** list, click the drive or folder of the location to which you want to save the file.
3. In the **File name** box, enter a name for the file, and then click **Save**.

Tip:

You can see the full path of the saved project file on the "Study" entity. (See also About Study Parameters (p. 415).)

See Also:

- Save an Exact Copy of the Simulation Project; Including the Simulation Results (p. 853)
- Save a Copy of Only the Simulation Settings; Not the Simulation Results (p. 854)
- Save a Copy of a Partially Processed Simulation for Restart Purposes (p. 855)
- Save Over the Current Simulation Project (p. 857)
- Save an Image of the 3D View, Plot, or Histogram (p. 858)

3.3.6. Save an Exact Copy of the Simulation Project; Including the Simulation Results

1. From the **File** menu, click **Save As**.
2. From the **Save As** dialog, choose the first option, "**Save As a New Project and Copy Simulation Results**", and then click **OK**.
3. In the **Save in** list, click the drive or folder of the location to which you want to save the file.

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

4. In the **File name** box, enter a name for the file, and then click **Save**.

Tip:

You can see the full path of the saved project file on the "Study" entity. (See also About Study Parameters (p. 415).)

See Also:

- Save a New Simulation Project (p. 853)
- Save a Copy of Only the Simulation Settings; Not the Simulation Results (p. 854)
- Save a Copy of a Partially Processed Simulation for Restart Purposes (p. 855)
- Save Over the Current Simulation Project (p. 857)
- Save an Image of the 3D View, Plot, or Histogram (p. 858)

3.3.7. Save a Copy of Only the Simulation Settings; Not the Simulation Results

---

Note:

If you do not want to save the results of your simulation, but you do want to reuse your setup parameters as a basis for your next analysis, you can also choose to delete the simulation results. (See also Delete the Results of a Simulation (But Keep the Setup) (p. 868).)

---

1. From the **File** menu, click **Save As**.
2. From the **Save As** dialog, choose the second option, "**Save As a New Project but DON'T Copy Simulation Results**," and then click **OK**.
3. In the **Save in** list, click the drive or folder of the location to which you want to save the file.
4. In the **File name** box, enter a name for the file, and then click **Save**.

Tip:

You can see the full path of the saved project file on the "Study" entity. (See also About Study Parameters (p. 415).)

See Also:

- Save a New Simulation Project (p. 853)
- Save an Exact Copy of the Simulation Project; Including the Simulation Results (p. 853)
- Save a Copy of a Partially Processed Simulation for Restart Purposes (p. 855)
- Save Over the Current Simulation Project (p. 857)

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

- Save an Image of the 3D View, Plot, or Histogram (p. 858)

3.3.8. Save a Copy of a Partially Processed Simulation for Restart Purposes

This procedure is useful in situations where you will be reusing the beginning part of a processed simulation as a starting place for other, new simulations. This saves you from having to repeat the beginning portions of a simulation over and over, conserving processing time. Some examples of when this might be useful include:

- Pre-filling a large bin or container with particles
- Changing-out boundaries in the latter part of a long chute or conveyor system

---

Important:

Because you are saving a copy of an already processed simulation, you will lose the ability to see time step files within the copy prior to the point you have saved it, and the processing time will be reset to zero (0).

Important:

When saving a simulation as a restart, existing contacts and forces are not stored. When the new, restarted simulation begins, new contacts will be identified, and the related forces will be calculated. Therefore, the initial results of a restart case may differ from those obtained at the saving point in the original case. This is especially true for history-dependent contact models, such as the Hysteretic Linear Spring (HLS).

REQUIREMENTS FOR SAVING A COPY OF A SIMULATION FOR RESTART PURPOSES

The main requirement for this procedure is that there must be particles released into the simulation at the time step you select when the copy is made. If the selected time step is zero or is at a point prior to when particles are released into the domain, no restart copy will be saved.

In this version of Rocky, you can save for restart purposes a partially processed copy of nearly any type of simulation, including those involving 1-Way LBM CFD Coupling, and all kinds of 1- and 2-Way Fluent coupling. (See also Set or Modify Fluid and/or Air Flow Properties (p. 757).)

LIMITATIONS TO SAVING A COPY OF A SIMULATION FOR RESTART PURPOSES

Although Rocky allows you to save for restart purposes a simulation with breakage enabled, it is important to know that any broken fragments (p. 1393) that are in the simulation at the time you choose to save a copy of it will not be saved in the copy. (See also Enable Particle Breakage Calculations (p. 397).)

In addition, if you have chosen to use parametric expressions in certain **Start Time** and **Stop Time** fields, only the resulting value and not the variables and/or mathematical functions will be retained

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

in the copy you have saved for restart purposes. (See also I get a "Links removed" message when I save my project for restart purposes (p. 1381).)

Tip:

Not all parameters in Rocky will be editable in your simulation copy. To learn more about what can and cannot be edited, including special considerations for Particle Inputs and Motion Frames, see About Changing Geometries and/or Particles in a Partially Processed Simulation (p. 847).

TO SAVE A COPY OF A PARTIALLY PROCESSED SIMULATION FOR RESTART PURPOSES:

1. From the **Time** toolbar, ensure that the current time step is set to the point at which you want the new copy to begin. (See also Use the Time Toolbar to Change the Current time step (p. 1280).) **Note**: There must be particles released into the domain at the time step you select otherwise a copy will not be saved.
2. From the **File** menu, click **Save project as…**.
3. From the **Save Options** dialog, choose the third option, "**Save as a New Project for Restart**", and then click **OK**. **Note**: This dialog will only appear if your simulation has been processed far enough to have results.
4. In the **Save File** dialog, select the folder in which you want to save the project copy.
5. In the **File name** box, enter a name for the new file, and then click **Save**.

Note: When you open the copy, the beginning time step will be reset to zero (0) but the simulation will start from the output file you choose in step #1.

Tips:

- Before you process your simulation copy, you may want to first Change Geometries and/or Particles in a Partially Processed Simulation (p. 847).
- You can see the full path of the saved project file on the "Study" entity. (See also About Study Parameters (p. 415).)
- To see a walk-through example of saving a copy of a Rocky project for restart purposes, refer to the following Workshop: Tutorial - Fluidized Bed in the *Rocky Tutorial Guide*

See Also:

- About Changing Geometries and/or Particles in a Partially Processed Simulation (p. 847)
- Save a New Simulation Project (p. 853)
- Save an Exact Copy of the Simulation Project; Including the Simulation Results (p. 853)
- Save a Copy of Only the Simulation Settings; Not the Simulation Results (p. 854)
- Save Over the Current Simulation Project (p. 857)
- Save an Image of the 3D View, Plot, or Histogram (p. 858)

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

3.3.9. Save Over the Current Simulation Project

- From the **File** menu, click **Save**.

Tip:

You can see the full path of the saved project file on the "Study" entity. (See also About Study Parameters (p. 415).)

See Also:

- Save a New Simulation Project (p. 853)
- Save an Exact Copy of the Simulation Project; Including the Simulation Results (p. 853)
- Save a Copy of Only the Simulation Settings; Not the Simulation Results (p. 854)
- Save a Copy of a Partially Processed Simulation for Restart Purposes (p. 855)
- Save Over the Current Simulation Project (p. 857)
- Save an Image of the 3D View, Plot, or Histogram (p. 858)

3.3.10. Save Custom Properties and Curves into projects

When adding a custom property or a custom curve, you can select in the **Scope** option whether you want to save it in the project, so that this curve or property is sent together when sharing the project, or save it only locally.

Figure 3.223: Save Custom Properties In the Project or Locally

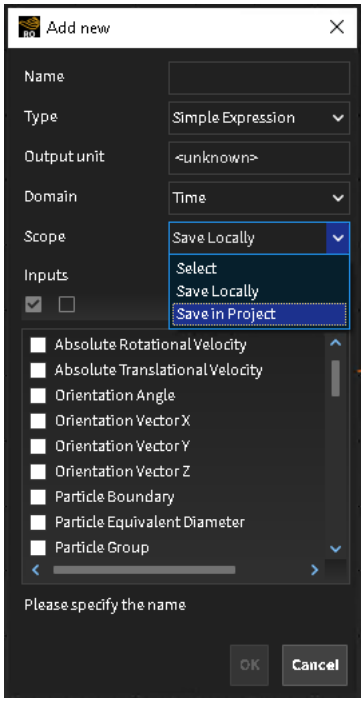
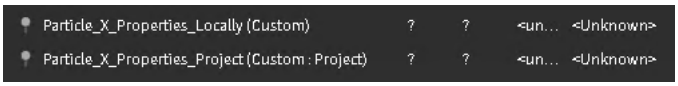


Figure 3.224: Custom Properties saved In the Project and Locally



3.3.11. Save an Image of the Rocky Window

1. In the Workspace (p. 1396), select the 3D View, Motion Preview, Particles Details, Plot, or Histogram window for which you want to save an image.
2. From the **Window** menu, click **Save Image**. The **Snapshot** dialog appears.
3. From the **Snapshot** dialog, enter a folder location, **File Name** and **Save as Type** information, and then click **Save**. The image is saved to the location you chose and Rocky asks you whether you want to open the snapshot.

Tips:

- If you want more control over the dimensions of your image, export your image from the Window Editors panel. (See also Save a Snapshot of the Rocky Window (p. 1163).)
- To make a copy the image instead of saving it, right-click the area within your window, point to **Copy**, and then click **Image**. A copy of the image will be placed on the clipboard for you to paste into an image editor or other program.

See Also:

- Save a New Simulation Project (p. 853)
- Save an Exact Copy of the Simulation Project; Including the Simulation Results (p. 853)
- Save a Copy of Only the Simulation Settings; Not the Simulation Results (p. 854)
- Save a Copy of a Partially Processed Simulation for Restart Purposes (p. 855)
- Save Over the Current Simulation Project (p. 857)

3.4. Verifying the Simulation Summary

Before you process a simulation, it is a good idea to check the key criteria provided on the **Simulation Summary** dialog. Doing this can help you catch data entry errors, and can help you get a rough estimate of how long the simulation might take to process.

At any point, you may also check the **Summary** tab on the Solver entity to gain access to the same information. From here, you can also see the Rocky version and simulation save dates for the project.

What would you like to do?

- Learn more About the Simulation Summary (p. 859)
- Use the Simulation Summary Dialog to Verify Key Simulation Criteria (p. 865)
- Use the Solver Summary Tab to Verify Key Simulation Criteria (p. 866)

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

- Use the Solver Summary Tab to Verify the Simulation Version (p. 866)

See Also:

- Set Simulation Parameters (p. 414)

3.4.1. About the Simulation Summary

Checking the key criteria listed on the **Simulation Summary** dialog (or tab) before you get too far into processing your simulation can help you catch data entry errors, which has the potential of saving you time in processing your simulation.

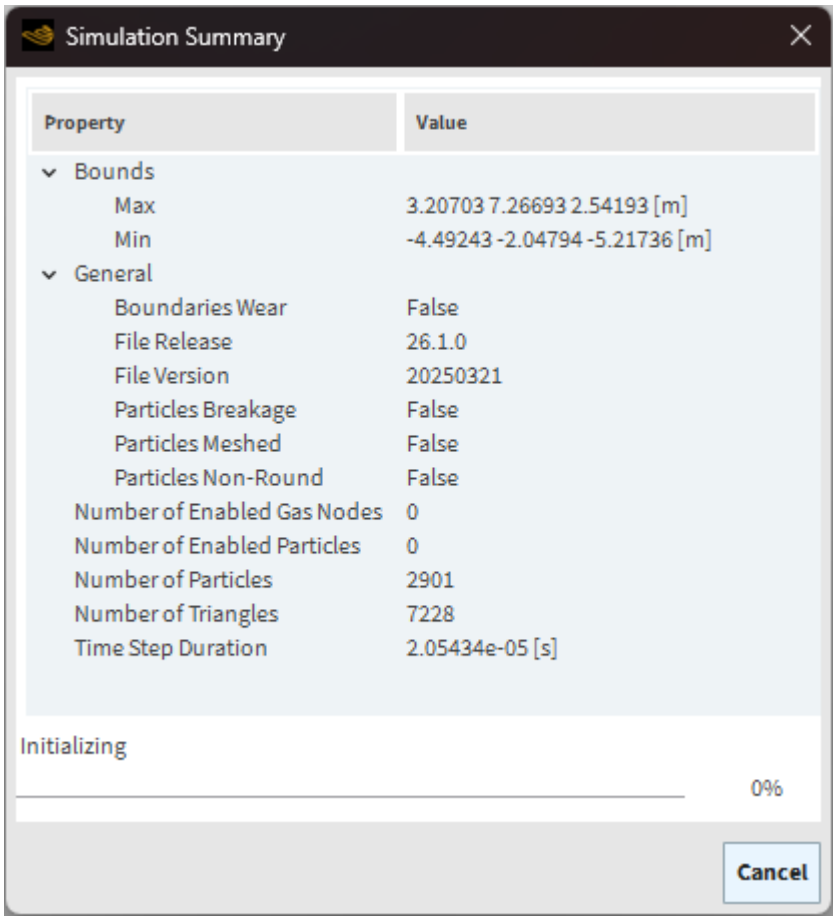
The Simulation Summary dialog automatically appears whenever you start a new simulation (see also Start Processing a Simulation from the Beginning (p. 913)). You can also view similar summary information after processing has started from the Solver entity. No matter whether processing has started or not, the Summary tab is also where you can find valuable timestamp information for the project, such as in what version of Rocky the project was created or last saved.

If you see information on the Summary that indicates that there might be an error in your setup-for example **Number of Particles** is higher than expected, **Particles Breakage** is turned on (marked as **True**) by mistake, and so on-you can either click **Cancel** on the dialog, which will stop the simulation from initialization; or if the dialog has already closed, you can click the **Stop** button on the **Solver** tab in the **Data Editors** panel. Both actions will stop the simulation from processing so that you can review and fix your setup information before continuing.

Besides helping you pin-point potential errors, the Summary information can also act as an indicator for how long it might take your simulation to process. While a higher **Time Step Duration** value will typically result in a faster simulation, for all other properties provided on the Simulation Summary, higher numbers and **True** values will generally result in longer simulation processing times. (See also Best Practices for Faster Processing (p. 68)). Tip:To get particle count estimate without having to start processing the simulation, you can also view the information on the **Info** tab of the Particle Inputs entity at any time during your project setup. For single-sized Particle sets, the counts provided on the Info tab will exactly match those provided on the Simulation Summary. For Particle sets with size distributions, the counts provided are estimates only and will differ from those on the Simulation Summary. (See also About the Info Tab (p. 83)). Use the figures and table below to understand the various types of summary information provided for simulations.

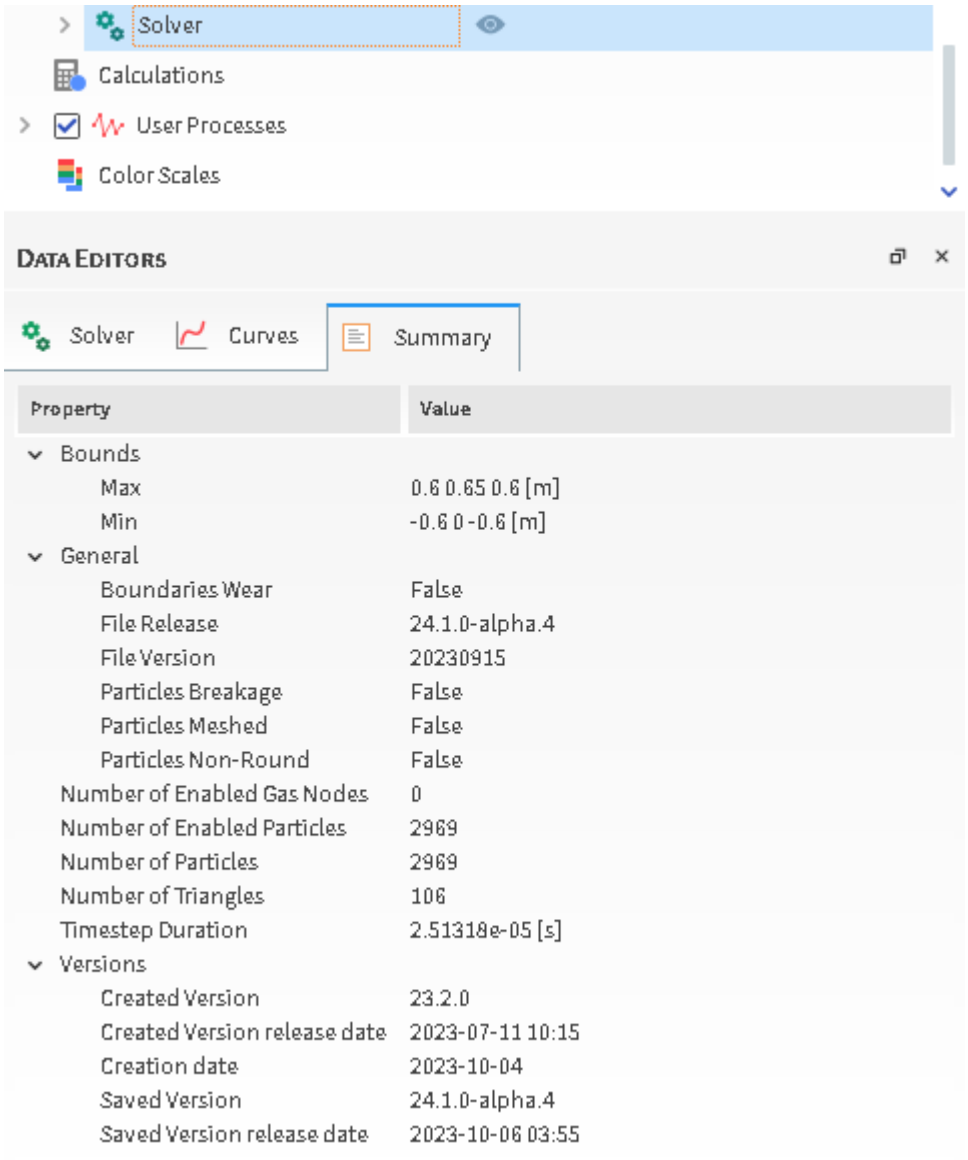
Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

Figure 3.225: Simulation Summary dialog



Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

Figure 3.226: Solver, Summary tab on the Data Editors panel before processing



Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

Figure 3.227: Solver, Summary tab on the Data Editors panel during or after processing

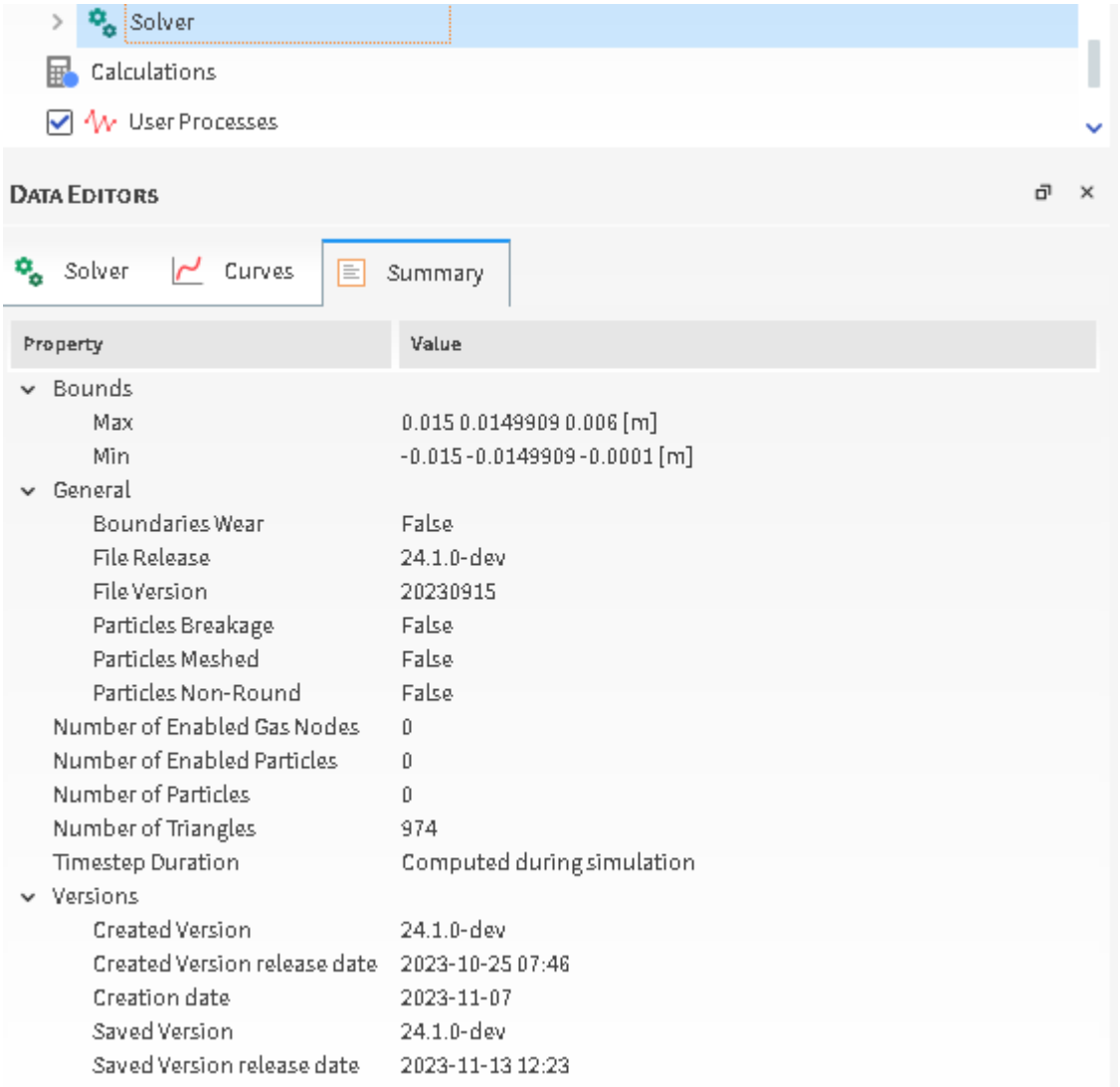


Table 1: Simulation Summary dialog and tab options

Property	Description
Bounds	
Max	The maximum coordinates for which geometry triangles are drawn. If **Set as Boundary Limits** is selected, once a particle reaches these limits, it will be removed from the calculations and will will disappear from any 3D Views. (This is done to save on processing power). (See also About Domain Settings Parameters (p. 452)). **Note:** Geometry limitations, including periodic domains, will also affect when particles enter and leave a simulation.

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

Property	Description
Min	The minimum coordinates for which geometry triangles are drawn. If **Set as Boundary Limits** is selected, once a particle reaches these limits, it will be removed from the calculations and will disappear from any 3D Views. (This is done to save on processing power). (See also About Domain Settings Parameters (p. 452)). **Note:** Geometry limitations, including periodic domains, will also affect when particles enter and leave a simulation.
General	
Boundaries Wear	Indicates whether surface wear modification calculations are enabled (True) or not (False). (See also View Surface Wear Modification on an Imported Geometry (p. 379)). **Note:** Surface wear modification calculations take additional time to process.
File Release	Number used to identify the name of the released Rocky version used to process the simulation project.
File Version	Number used to identify the release date of the Rocky solver version that was used to process the original simulation project. Due to the way Rocky is programmed, this can be a different date than the date the Rocky program itself was released; knowing this date can be useful to Rocky support when troubleshooting simulation issues. The format of the date is YYYYMMDD with YYYY representing the four-digit year, MM representing the two-digit month, and DD representing the two digit day.
Particles Breakage	Indicates whether particle breakage calculations are enabled (True) or not (False). (See also Enable Particle Breakage Calculations (p. 397)). **Note:** Breakage calculations take additional time to process.
Particles Meshed	Indicates whether any of the particle sets (p. 1393) are composed of multiple elements (True) or not (False). (See also About Adding and Editing Particle Sets (p. 602)).

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

Property	Description
Particles Non-Round	Indicates whether shaped particles (p. 1395) are simulated (True) or not (False). **Note:** Shaped particles take additional time to process when compared to Spheres.
Number of Elements	For projects in which there are particle sets (p. 1393) composed of multiple elements (see also About Adding and Editing Particle Sets (p. 602)), this lists the total number of elements (p. 1390) that will appear in the simulation, from start to finish. For projects including particle sets (p. 1393) composed of both single and multiple elements, each single-element particle will count as zero elements.
Number of Enabled Elements	For projects in which there are particle sets (p. 1393) composed of multiple elements (see also About Adding and Editing Particle Sets (p. 602)), this lists the total number of elements (p. 1390) already present in the simulation at simulation onset. For most projects, this will be zero. Values higher than zero will be possible only for partially processed projects that have been saved for restart purposes. (See also Save a Copy of a Partially Processed Simulation for Restart Purposes (p. 855)).
Number of Enabled Gas Nodes	When 1-Way LBM CFD Coupling calculations are turned on, this is the number of air nodes that will be used during the simulation. The number is calculated based on air flow cell size and air flow boundary limits. (See also Use the 1-Way LBM Method (p. 770)).
Number of Enabled Particles	Total number of particles already present in the simulation at simulation onset. For most projects, this will be zero. Values higher than zero will be possible only for partially processed projects that have been saved for restart purposes. (See also Save a Copy of a Partially Processed Simulation for Restart Purposes (p. 855)).
Number of Particles	Total number of whole particles that will appear in the simulation, from start to finish. The number is calculated based on particle size, mass flow rate, and density values, as well as inlet release times. (See also Add and Edit Particle Inputs (p. 708)). **Note:** This value includes whole particles only and does not take into account particle fragments (p. 1393) nor elements (p. 1390). **Tip:** You can get a more detailed estimate of particle count at any time during simulation setup by viewing the **Info** tab for the Particle Input entities. (See also About the Info Tab (p. 83)).

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

Property	Description
Number of Triangles	Total number of geometry triangles that will be used in the simulation. The number is either obtained from imported CAD geometry files or is calculated based on the **Triangle Size** value specified in the various settings dialogs for geometries.
Time Step Duration	This critical value for ensuring the stability and accuracy of the simulation is automatically calculated based upon the particle size, density, bulk Young's Modulus, and restitution coefficients used in the simulation. **Note:** This number will change based upon simulation parameters.
Versions (Summary Tab only)	
Created Version	The Rocky version that was used to create (first save) the original project.
Created Version release date	The release date and time of the Rocky version that was used to create (first save) the original project.
Creation date	The date that the original project was created (first saved).
Saved Version	The Rocky version that was used to save the latest version of the project.
Saved Version release date	The release date and time of the Rocky version that was used to save the latest version of the project.

What would you like to do?

- Use the Simulation Summary Dialog to Verify Key Simulation Criteria (p. 865)
- Use the Solver Summary Tab to Verify Key Simulation Criteria (p. 866)
- Use the Solver Summary Tab to Verify the Simulation Version (p. 866)

See Also:

- Set Simulation Parameters (p. 414)

3.4.2. Use the Simulation Summary Dialog to Verify Key Simulation Criteria

1. Ensure that you have set up the minimum requirements for processing. (See also Setting Up a Simulation (p. 411).) Tip:Before processing, you can get an estimate on how many particles might be released into the simulation by viewing the **Info** tab for the Particle Input entities. (See also About the Info Tab (p. 83).)

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

2. From the **Data** panel, select **Solver** and then from the **Data Editors** panel, click **Start**. The **Simulation Summary** dialog appears, which you can review while the simulation is automatically initialized.

Tip:

The **Simulation Summary** dialog will close automatically when the simulation is fully initialized and begins processing. If the dialog closes too quickly for you to review it, you may review the same information on the **Summary** tab of the main **Solver** entity.

See Also:

- About the Simulation Summary (p. 859)
- Processing a Simulation (p. 907)

3.4.3. Use the Solver Summary Tab to Verify Key Simulation Criteria

---

Tip:

This procedure can be followed only after a simulation has begun processing.

---

1. From the **Data** panel, select **Solver** and then from the **Data Editors** panel, select the **Summary** tab.
2. Review the information provided.

See Also:

- About the Simulation Summary (p. 859)
- Processing a Simulation (p. 907)

3.4.4. Use the Solver Summary Tab to Verify the Simulation Version

This task is especially useful when you are trying to determine what version of your project is the latest, or when you need to know with what version of Rocky the original project was created. Knowing the latter can help you determine what Rocky features and functionality are available for the project.

Tip:

This procedure can be followed at any point during or after simulation setup and processing.

---

1. From the **Data** panel, select **Solver** and then from the **Data Editors** panel, select the **Summary** tab.
2. Under **Versions**, review the information provided.

See Also:

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

- About the Simulation Summary (p. 859)
- Processing a Simulation (p. 907)

3.5. Changing the Setup After Processing

Once you have run your simulation, you may want to change out geometries or reset certain values before running the simulation again. In Rocky, there are three main ways to do this:

1. **Create a Copy for Restart**: In this scenario, you would first select (from the **Time** toolbar) which time step you want to use as the starting point for your copy, then you would Save a Copy of a Partially Processed Simulation for Restart Purposes (p. 855) at that point. Within the new copy-whose time has been reset to "0" seconds but whose original results are still visible-you would then change out only certain setup parameters such as geometry files or particle sets (see also About Changing Geometries and/or Particles in a Partially Processed Simulation (p. 847)), and then restart processing within the copy. This is useful when you want to keep the beginning results of your original simulation, but want to change some of the parameters for the latter portion of processing.

2. **Save a Copy of the Setup**: In this scenario, you would Save a Copy of Only the Simulation Settings; Not the Simulation Results (p. 854), change out any setup parameters you like in the new copy, and then process the new copy. This is useful when you want to keep the results of your simulation but want to reuse the setup parameters in a different but similar project.

3. **Delete Your Simulation Results**: In this scenario, you would first Delete the Results of a Simulation (But Keep the Setup) (p. 868), change out any setup parameters you like, and then start processing the project from the beginning. This is useful when you know you do not want to keep your simulation results but do want to reuse the setup parameters in your next analysis.

In both the latter options, you are able to Set Your Simulation Parameters (p. 414) as usual prior to running your simulation again. In only the first option are you limited in what setup parameters you can change.

What would you like to do?

- Creating a Simulation Project (p. 411)
- Save a Simulation Project (p. 852)
- Set Simulation Parameters (p. 414)
- Verify Key Simulation Criteria in the Summary (p. 858)
- Delete the Results of a Simulation (But Keep the Setup) (p. 868)

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

3.6. Deleting the Simulation Results while keeping the setup

If you know that you do not want to keep the results of your simulation, but you do want to reuse your setup parameters as a basis for your next analysis, you can choose to delete the simulation results.

Note:

You can also keep the results and start a new simulation using a copy of the setup parameters. (See also Save a Copy of Only the Simulation Settings; Not the Simulation Results (p. 854).)

3.6.1. To delete the results of a simulation (but keep the setup):

1. With your processed simulation stopped (see also Stop a Simulation from Processing (p. 915)) or completed, change any setup parameter, such as **Simulation Duration**. A new dialog appears.
2. From the **Project already has simulated results** dialog, click **Yes**. The results of your simulation are deleted and your setup parameters are fully editable.

See Also:

- Setting Up a Simulation (p. 411)
- Save a Simulation Project (p. 852)
- Set Simulation Parameters (p. 414)

3.7. Discovering Rocky Embedded Modules

Several Rocky features are provided via Modules, including Collision and Particle Statistics. Because these Modules are provided by default with your Rocky installation, they are referred to as **Embedded Modules**.

The Embedded Modules included by default with Rocky software are in following sections.

To also see the documentation for the Rocky External Modules (Ready-to-use Modules) available for download from the Ansys Customer Portal, access the Rocky Modules Manual.

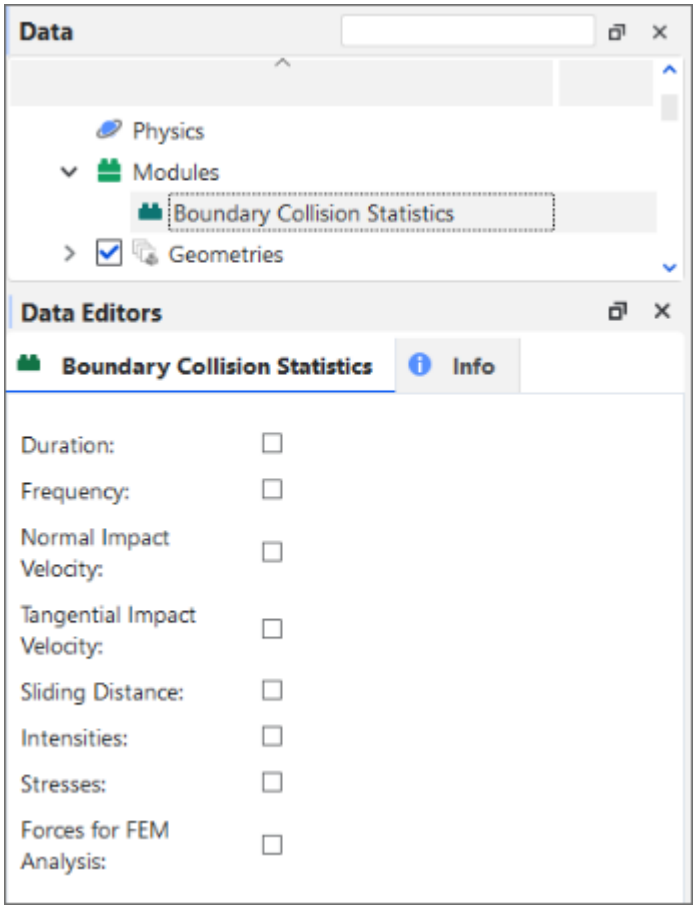
3.7.1. Boundary Collision Statistics Module

The Boundary Collision Statistics module enables the collection of boundary-related collision data, such as collision frequency, intensities, and impact velocities.

MODULE OPTIONS

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

Figure 3.228: Options in the Data Editors panel when the Boundary Collision Statistics Module is enabled



When the Boundary Collision Statistics Module is enabled (Figure 1), you are able to select any of the following Properties:

- Duration
- Forces for FEM Analysis
- Frequency
- Intensities
- Normal Impact Velocity
- Sliding Distance
- Stresses
- Tangential Impact Velocity

After processing your simulation, specific Properties and Curves for the options you enabled will be available for the boundaries in your simulation.

What would you like to do next?

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

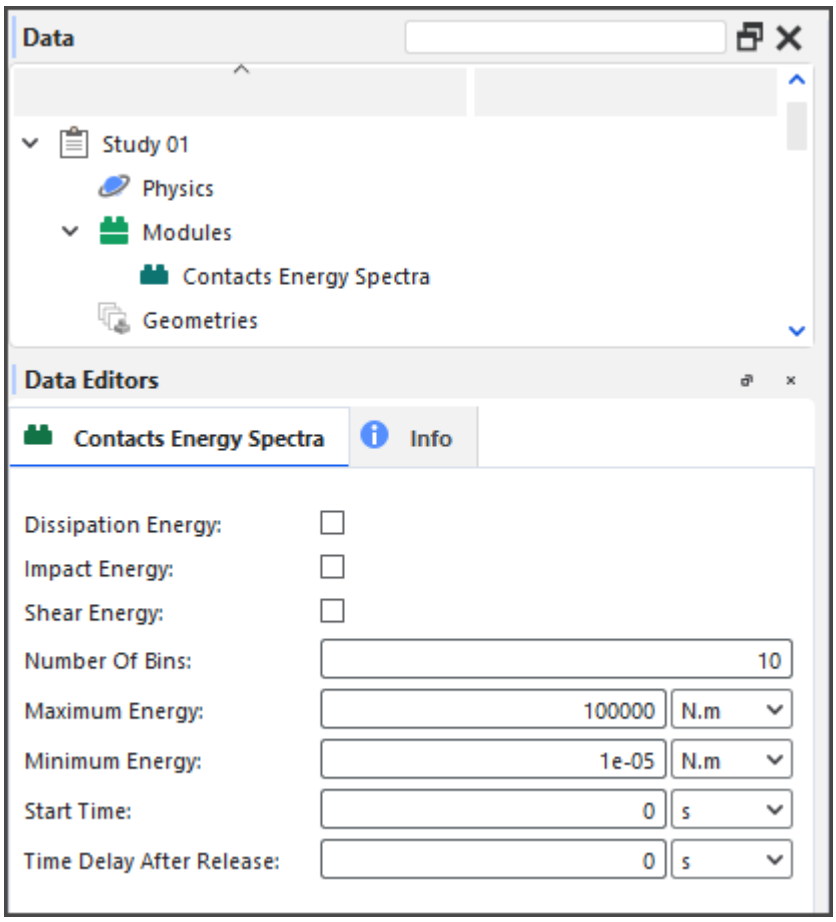
- Learn more About Collision Statistics for Boundaries (p. 312)
- Enable and View Collision Statistics for Boundaries (p. 318)
- Learn more About Collecting Data in Rocky (p. 306)

3.7.2. Contacts Energy Spectra Module

The **Contacts Energy Spectra** module enables the collection of different kinds of energy statistics per contact pair (particle group and/or geometry) and size, which can help you predict breakage and attrition rates for continuous processes such as grinding mills.

MODULES OPTIONS

Figure 3.229: Options in the Data Editors panel when the Contacts Energy Spectra Module is enabled



When the **Contacts Energy Spectra** Module is enabled (Figure 1), you can choose to collect one or more of three different types of collision energy-Dissipation Energy, Impact Energy, and/or Shear Energy-and define the limits for how the energy data is collected.

In this version of Rocky, you can also choose which particle group and geometry component you want to participate in energy spectra collection.

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

Figure 3.230: Additional module options for Particle groups when the Contacts Energy Spectra module is enabled

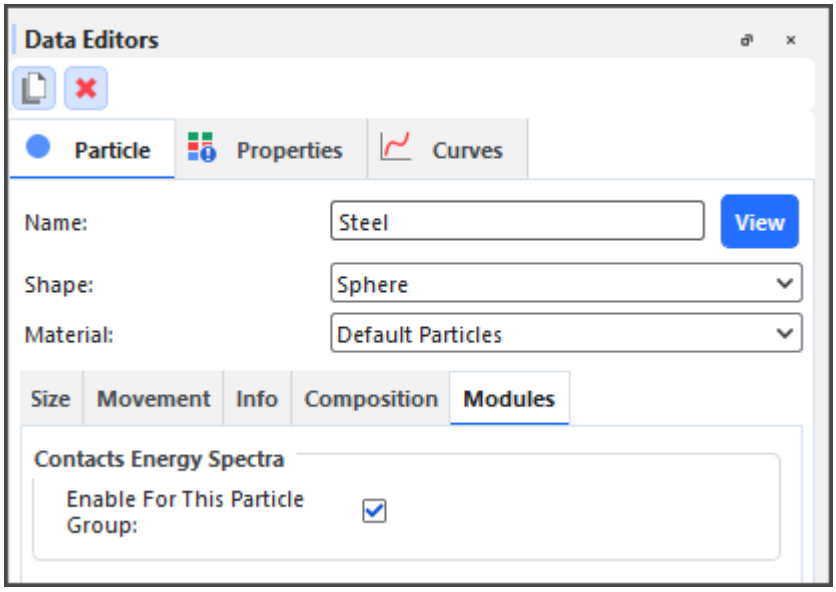
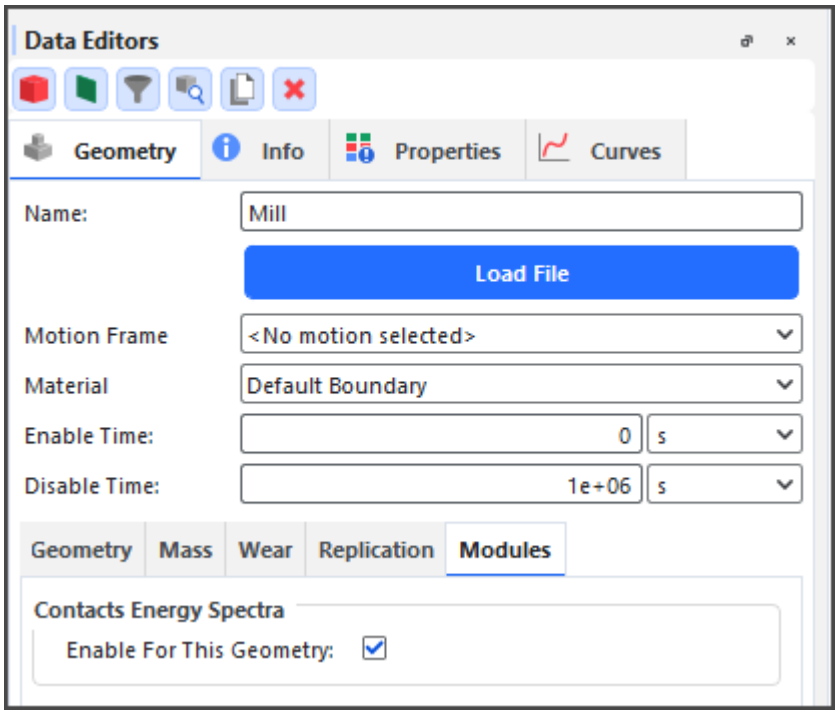


Figure 3.231: Additional module options for a geometry component when the Contacts Energy Spectra module is enabled



After processing your simulation, you can plot the resulting energy curves (Dissipation, Impact, and/or Shear) for each pair of particle-to-particle or particle-to-geometry contact types, each generated by the power, cumulative power, and collisions rate.

What would you like to do next?

- Learn more About Contacts Energy Spectra (p. 350)

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

- Enable and View Data for Contacts Energy Spectra (p. 367)
- Learn more About Collecting Data in Rocky (p. 306)

3.7.3. Contacts Overlap Monitor Module

When enabled, the **Contacts Overlap Monitor** module checks each contact (p. 1390) pair (particle-particle or particle-boundary) for the amount that they overlapped-the percentage of which is determined by the size of the smallest particle in the contact pair-and then raises a warning in the Simulation Log panel (Figure 1) if an overlap exceeds any of the three warning levels you define (Figure 2).

Figure 3.232: Example Contacts overlap message on the Simulation Log panel

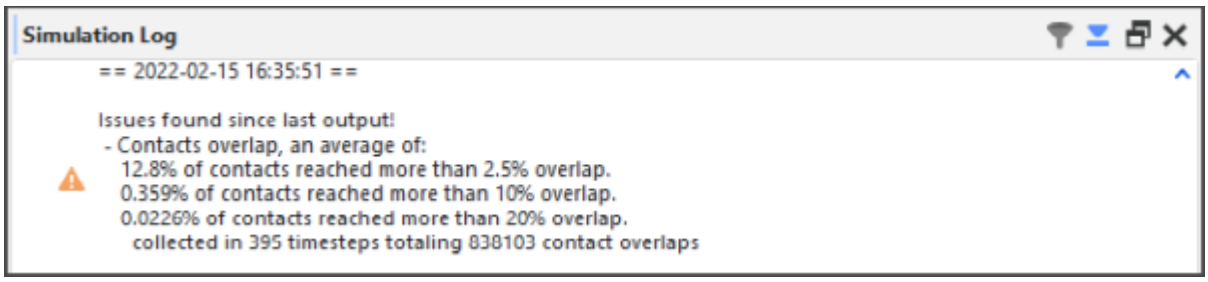
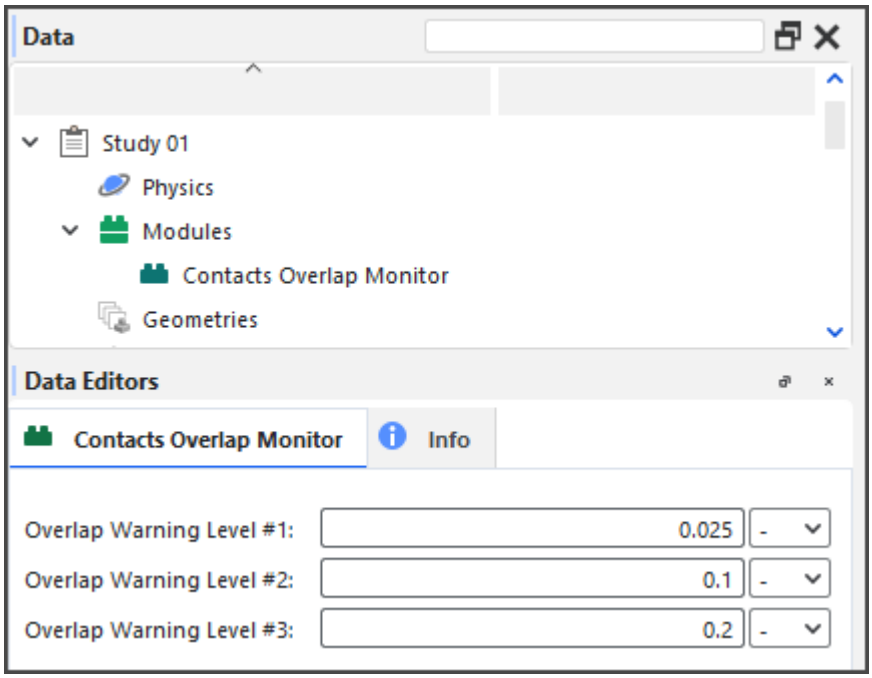


Figure 3.233: Options in the Data Editors panel when the Contacts Overlap Monitor Module is enabled



Monitoring your contacts for overlaps is important because Rocky uses the overlap value in order to compute collision forces. Therefore, large overlap values can lead to serious stability and accuracy issues in a simulation.

What would you like to do next?

- Learn more About Monitoring Overlaps (p. 309)

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

- Turn off the Overlap Monitor (p. 311)
- Learn more About Collecting Data in Rocky (p. 306)

See Also:

- About the Simulation Log Panel (p. 101)
- I get warnings or errors on my Simulation Log Panel (p. 1346)

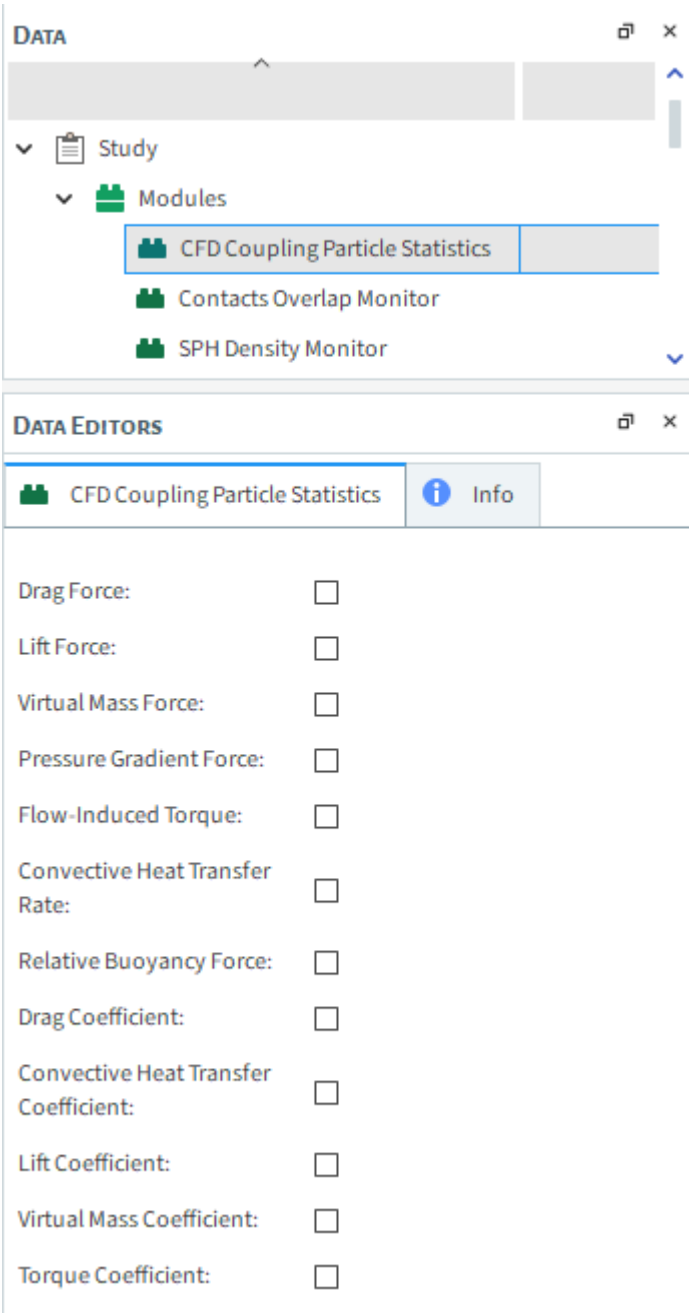
3.7.4. CFD Coupling Particle Statistics Module

The CFD Coupling Particle Statistics Module enables the collection of particle-fluid interactions, such as drag, lift, and virtual mass forces.

MODULE OPTIONS

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

Figure 3.234: Options in the Data Editors panel when the CFD Coupling Particle Statistics Module is enabled



When the CFD Coupling Particle Statistics Module is enabled (Figure 1), you are able to select any of the following Properties:

- Convective Heat Transfer Rate
- Drag Force
- Flow-Induced Torque
- Lift Force

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

- Pressure Gradient Force
- Virtual Mass Force
- Relative Buoyancy Force
- Drag Coefficient
- Convective Heat Transfer Coefficient
- Lift Coefficient
- Virtual Mass Coefficient
- Torque Coefficient

After processing your simulation, specific Properties for the options you enabled will be available for the particles in your CFD Coupling simulation.

What would you like to do next?

- Learn more About Fluid-Related Statistics for Particles (p. 335)
- Enable and View Fluid-Related Statistics for Particles (p. 349)
- Learn more About Collecting Data in Rocky (p. 306)

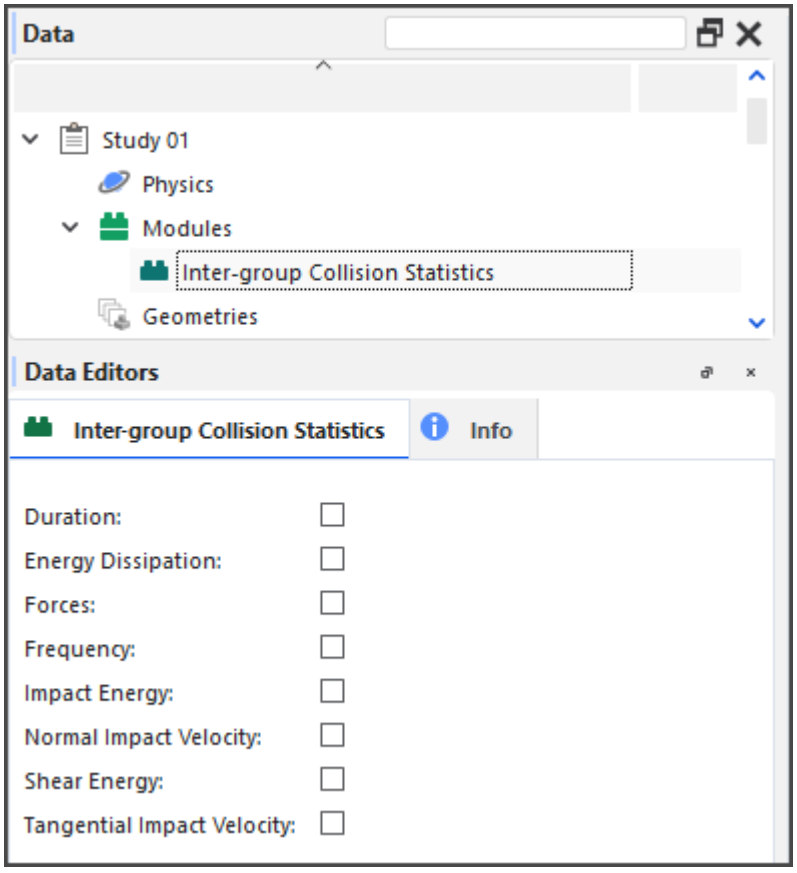
3.7.5. Inter-group Collision Statistics Module

The Inter-group Collision Statistics module enables the collection of collisions data for each particle-particle and particle-boundary pair group.

MODULE OPTIONS

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

Figure 3.235: Options in the Data Editors panel when the Inter-group Collision Statistics Module is enabled



When the Inter-group Collision Statistics Module is enabled, you are able to select the following Properties:

- Duration
- Energy Dissipation
- Forces
- Frequency
- Impact Energy
- Normal Impact Velocity
- Shear Energy
- Tangential Impact Velocity

In this version of Rocky, you can also choose which particle group (Figure 2) and geometry component (Figure 3) you want to participate in these collections.

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

Figure 3.236: Additional module options for Particle groups when the Inter-group Collision Statistics module is enabled

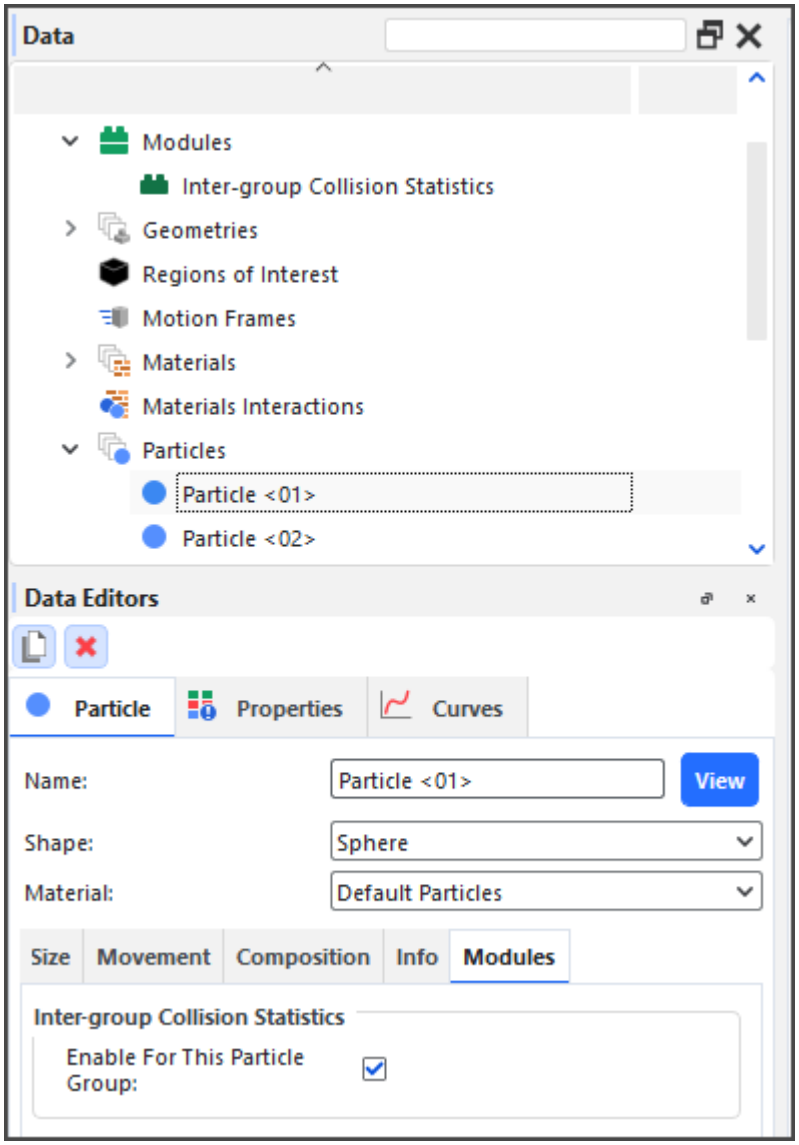
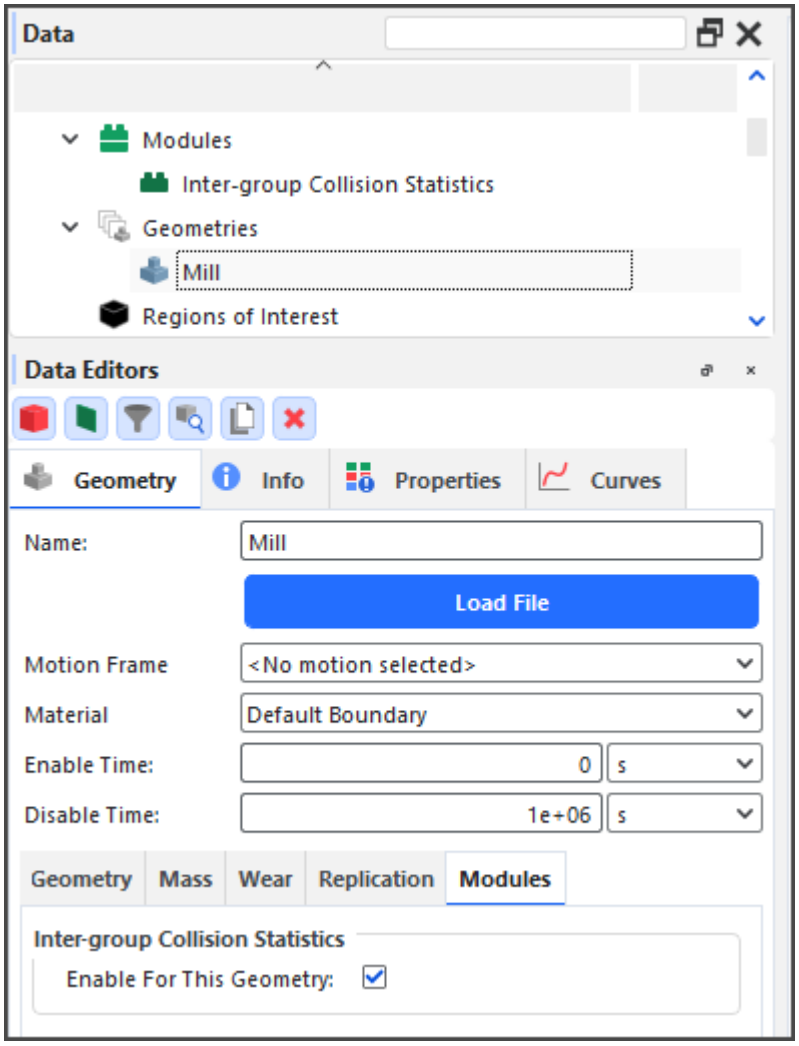


Figure 3.237: Additional module options for a geometry component when the Inter-group Collision Statistics module is enabled



After processing your simulation, specific Curves for the options you enabled will be available for the main Particles entity.

What would you like to do next?

- Learn more About Inter-group Collision Statistics (p. 319)
- Enable and View Inter-group Collision Statistics for Particles (p. 345)
- Learn more About Collecting Data in Rocky (p. 306)

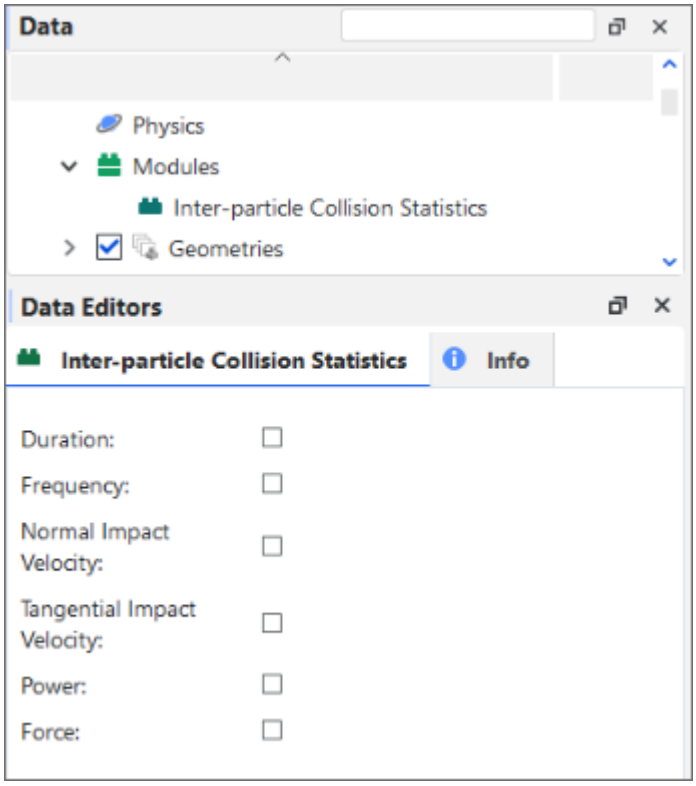
3.7.6. Inter-particle Collision Statistics Module

The Inter-particle Collision Statistics module enables the collection of collisions effects upon individual particles resulting from interactions with other particles and boundaries.

MODULE OPTIONS

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

Figure 3.238: Options in the Data Editors panel when the Inter-particle Collision Statistics Module is enabled



When the Inter-particle Collision Statistics Module is enabled, you are able to select any of the following Properties:

- Duration
- Force
- Frequency
- Normal Impact Velocity
- Power
- Tangential Impact Velocity

After processing your simulation, specific Properties and Curves for the options you enabled will be available for the main Particles entity.

What would you like to do next?

- Learn more About Inter-particle Collision Statistics (p. 331)
- Enable and View Collision Statistics for Particles Between Other Particles (Inter) (p. 348)
- Learn more About Collecting Data in Rocky (p. 306)

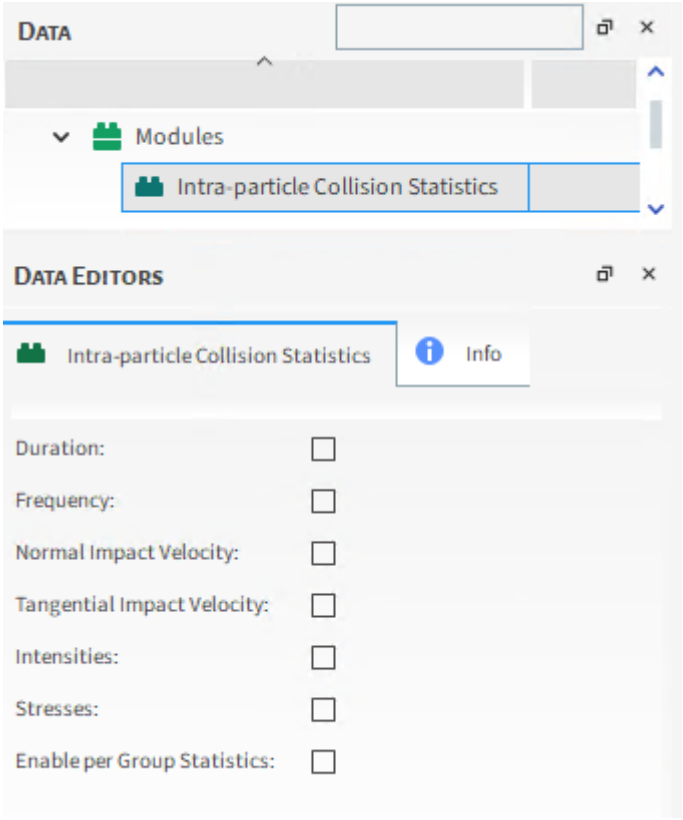
Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

3.7.7. Intra-particle Collision Statistics Module

The Intra-particle Collision Statistics module enables the particle-related collision data affecting the surfaces of a Particle set (p. 1393).

MODULE OPTIONS

Figure 3.239: Options in the Data Editors panel when the Intra-particle Collision Statistics Module is enabled



When the Intra-particle Collision Statistics Module is enabled, you are able to select any of the following Properties:

- Duration
- Frequency
- Normal Impact Velocity
- Tangential Impact Velocity
- Intensities
- Stresses
- Enable per Group Statistics

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

After processing your simulation, specific Properties for the options you enabled will be available for each individual Particle set. You can then choose to display this information graphically on the surface of a representative particle in the **Particles Details** window

What would you like to do next?

- Learn more About Intra-particle Collision Statistics (p. 324)
- Enable and View Collision Statistics for Particle Surfaces (Intra) (p. 346)
- Learn more About Collecting Data in Rocky (p. 306)

3.7.8. Multibody Dynamics FMU Coupling Module

The Multibody Dynamics FMU Coupling Module enables you to *import* FMU files into Rocky without needing to install external Rocky modules into compatible softwares.

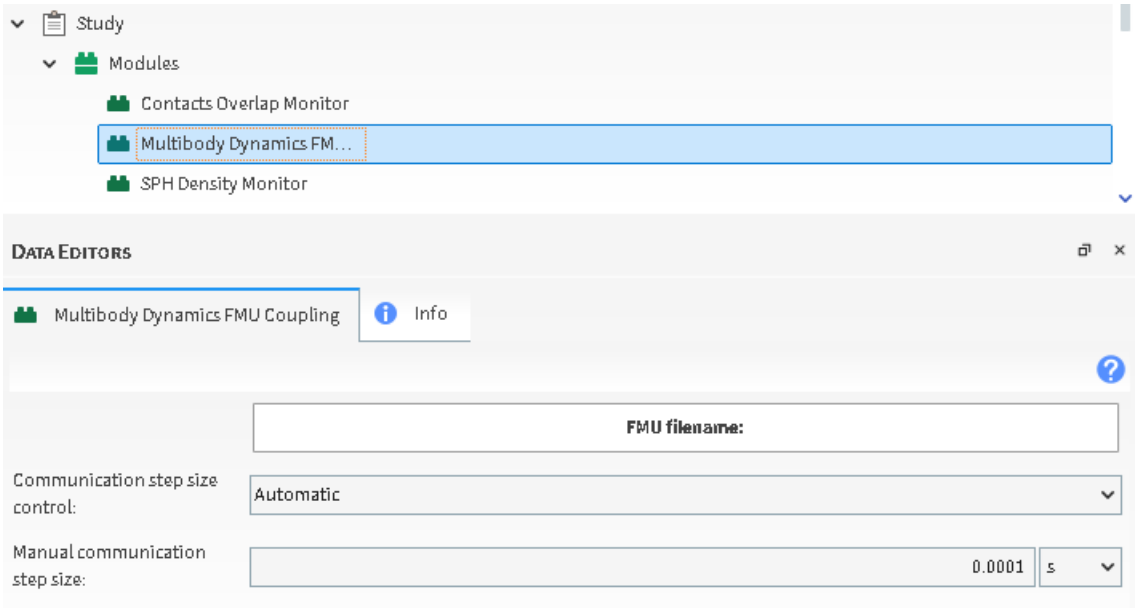
Note:

For Ansys Motion software and Adams, modules for these software are still needed to *export* the FMU files.

Similar to the Rocky support for the center of mass or the arrow (normal or not), that follow an internal motion frame (associated with a geometry), now both follow an external motion, as an FMU, through Multi-Body Dynamics (MBD) coupling with Ansys Motion.

MODULE OPTIONS

Figure 3.240: Options in the Data Editors panel when the Multibody Dynamics FMU Coupling is enabled



When the Multibody Dynamics FMU Coupling Module is enabled, you are able to import a FMU file through the **FMU filename:** button, and select the following properties:

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

- **Communication step size control:** configures the step size method for the communication between Rocky and the FMU file. Three different options can be selected:
  - Automatic: this method allows Rocky to automatically calculate a communication step size control.
  - Manual: this method requires a manual input from you. If the manual input value is smaller than Rocky's time step, it will be overruled by the one calculated by Rocky.
  - Rocky time step: this method considers the simulation time step calculated by Rocky to be the communication step size control.
- **Manual communication step size:** configures the step size of the manual communication method.

  Rage: [Positive Values]

  Note:

  This option is only used when the **Manual** communication step size control is selected.

See Also

- Rocky Simulation Entities that can be Affected by Modules (p. 223)

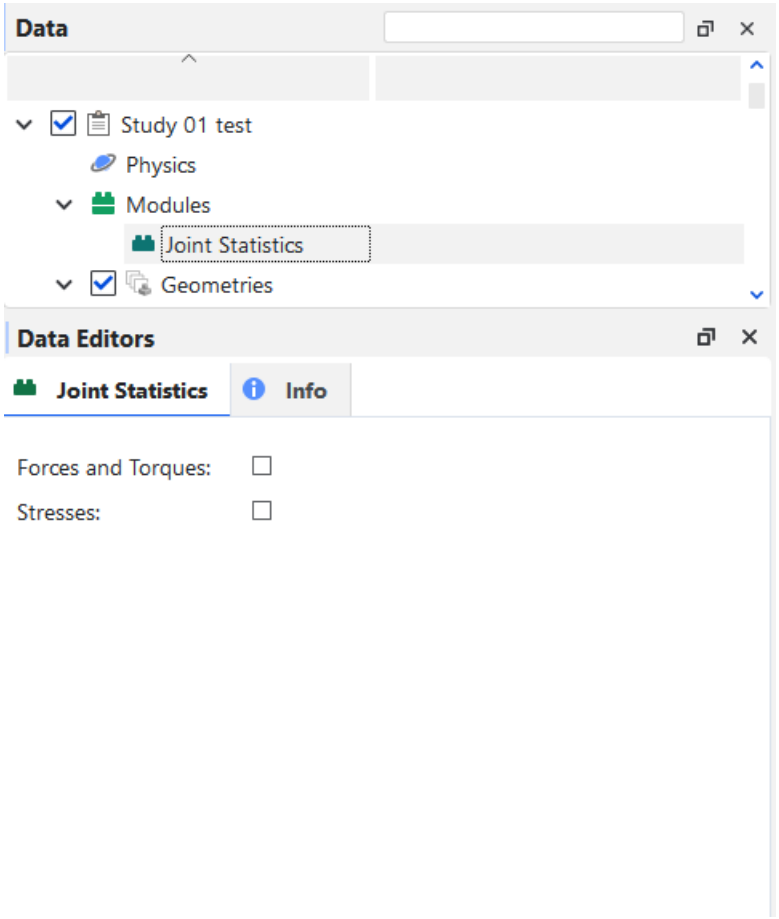
3.7.9. Joint Statistics Module

The Joint Statistics module enables new joint properties, with the purpose of generating new statistics data for analysis, such as **Stresses** and **Forces and Torques** calculations.

MODULE OPTIONS

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

Figure 3.241: Options in the Data Editors panel when the Joint Statistics Module is enabled



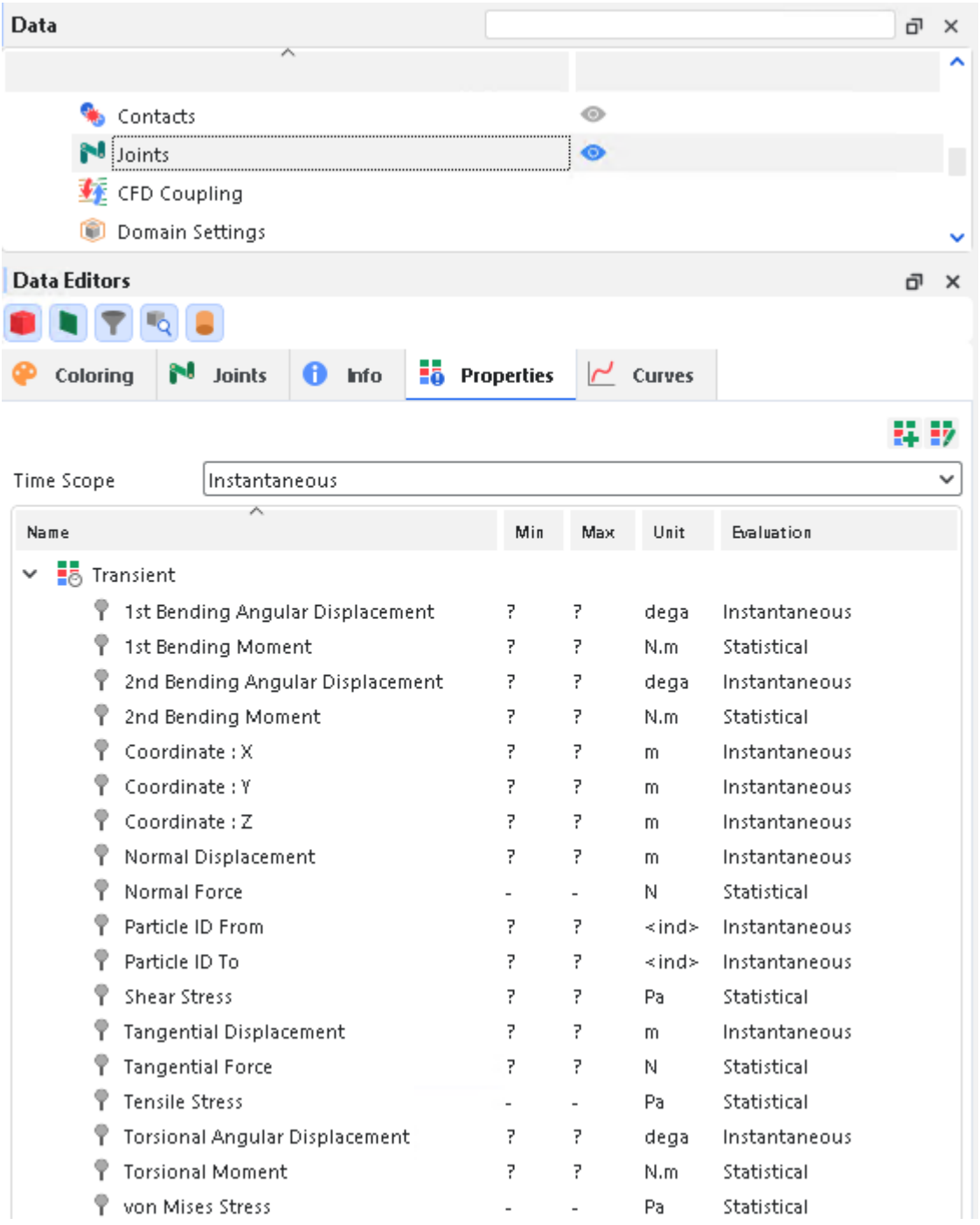
When the Joint Statistics Module is enabled (Figure 1), you are able to select any of the following Properties:

- Forces and Torques
- Stresses

This way, in the post-processing phase, it is possible to visualize the results of the parameters selected in Joints (p. 447) properties.

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

Figure 3.242: Statistic data in joint properties



See Also

- About Particle Breakage (p. 665)
- About Adding and Editing Particle Sets (p. 602)
- Rocky Simulation Entities that can be Affected by Modules (p. 223)

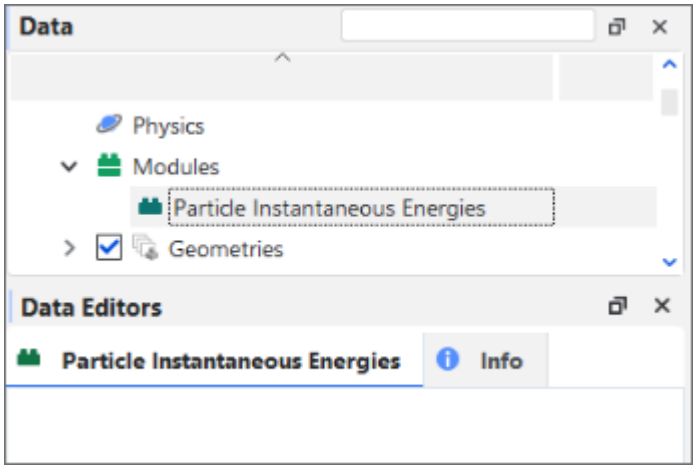
Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

3.7.10. About the Particle Instantaneous Energies Module

The Particle Instantaneous Energies module enables the collection of energies data related to the velocities and positions of each particle in the simulation. This data can then be used to calculate the kinetic and potential energies of each individual particle, which can be useful when performing global or partial energy balances in a simulation.

MODULE OPTIONS

Figure 3.243: There are no options in the Data Editors panel when the Particle Instantaneous Energies Module is enabled



When the Particle Instantaneous Energies Module is enabled, there are no properties or settings to enable.

After processing your simulation, specific Properties for the options you enabled will be available for the main Particles entity.

What would you like to do next?

- Enable and View Instantaneous Energies for Particles (p. 348)
- Learn more About Particle Instantaneous Energies (p. 341)
- Learn more About Collecting Data in Rocky (p. 306)

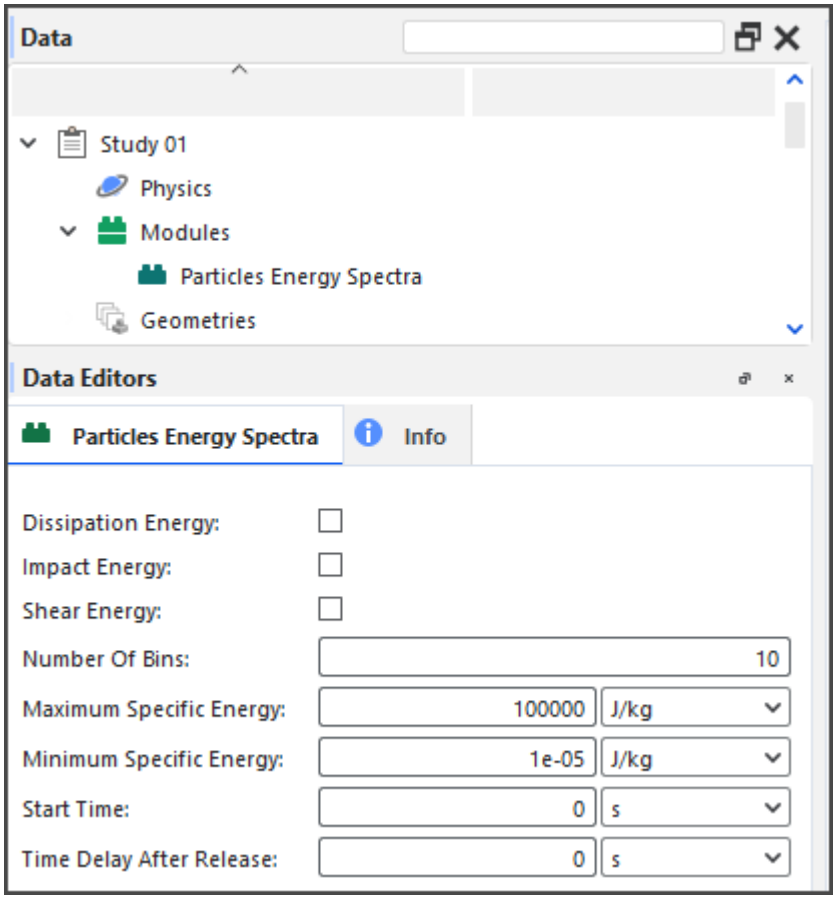
3.7.11. Particles Energy Spectra Module

The **Particles Energy Spectra** module enables the collection of different kinds of energy statistics per particle type and size category, which can help predict breakage and attrition rates for continuous processes such as grinding mills.

MODULE OPTIONS

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

Figure 3.244: Options in the Data Editors panel when the Particles Energy Spectra Module is enabled

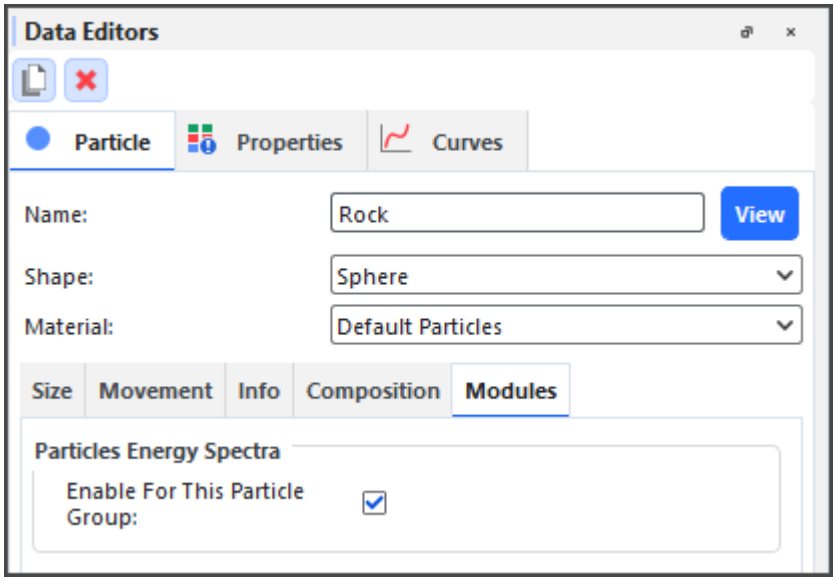


When the **Particles Energy Spectra** module is enabled (Figure 1), you can choose to collect one or more of three different types of collision energy-Dissipation Energy, Impact Energy, and/or Shear Energy-and define the limits for how the energy data is collected.

In this version of Rocky, you can also choose which particle group (Figure 2) you want to participate in energy spectra collection.

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

Figure 3.245: Additional module options for Particle groups when the Particles Energy Spectra module is enabled



After processing your simulation, you can plot the resulting Cumulative Specific Power curves for the type of energy you collected (Dissipation, Impact, and/or Shear) for each particle set, generated by each size of the particle size distribution, and separated by Specific Energy.

What would you like to do next?

- Learn more About Particles Energy Spectra (p. 358)
- Enable and View Data for Particles Energy Spectra (p. 369)
- Learn more About Collecting Data in Rocky (p. 306)

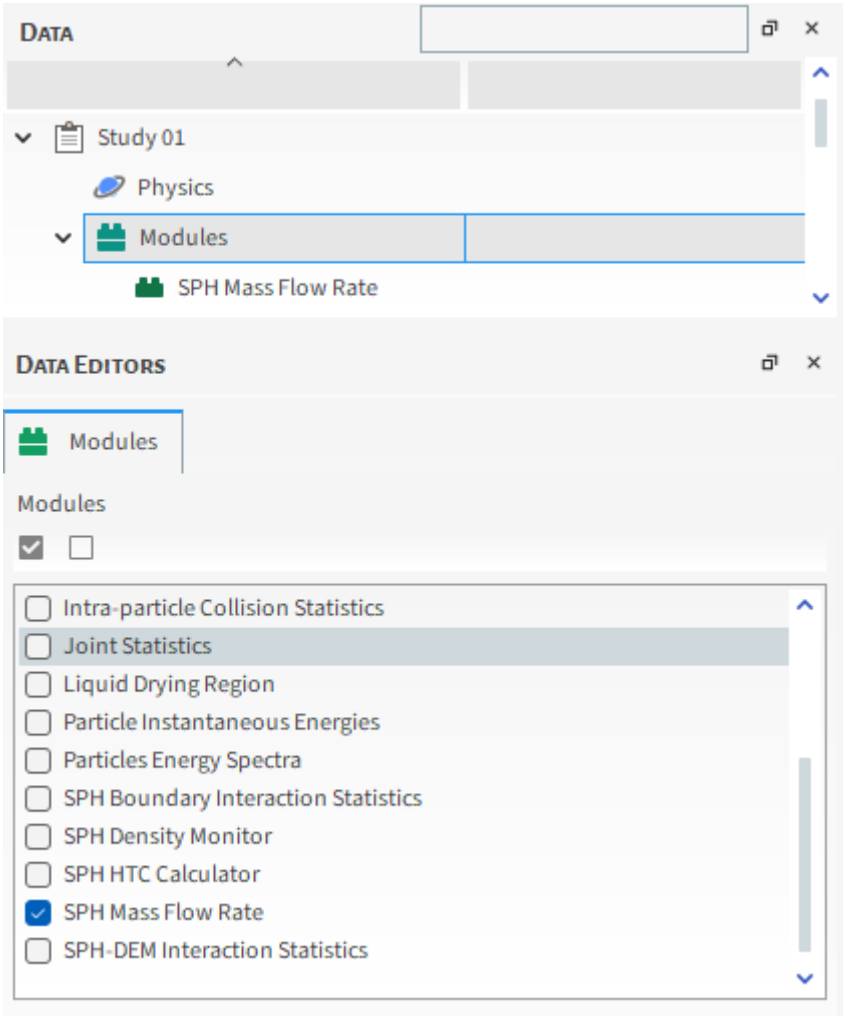
3.7.12. SPH Mass Flow Rate Module

The SPH Mass Flow Rate module is a flow meter that acts by calculating the SPH mass flow rate through a surface, enabling you to choose where you want to measure the mass flow rate.

MODULE OPTIONS

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

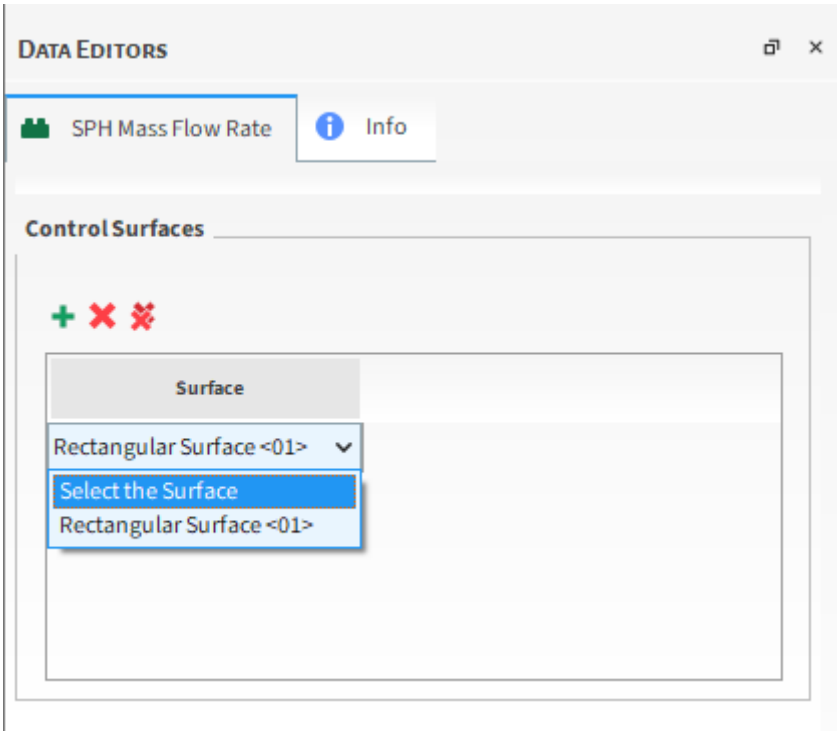
Figure 3.246: Options in the Data Editors panel when the SPH Mass Flow Rate Module is enabled



When the SPH Mass flow Rate Module is enabled (Figure 1), you are able to select one or more surfaces present in the simulation and measure the mass flow rate through them.

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

Figure 3.247: Geometries options in the Data Editors panel when the SPH Mass Flow Rate Module is enabled



After processing your simulation, specific mass flow rate curves will be available for the surfaces you selected through the module.

What would you like to do next?

- Learn more About the Filter User Process (p. 1202)
- Filter Views and Data with User Processes (p. 1174)
- Learn more About the CFD Coupling Particle Statistics Module (p. 873)

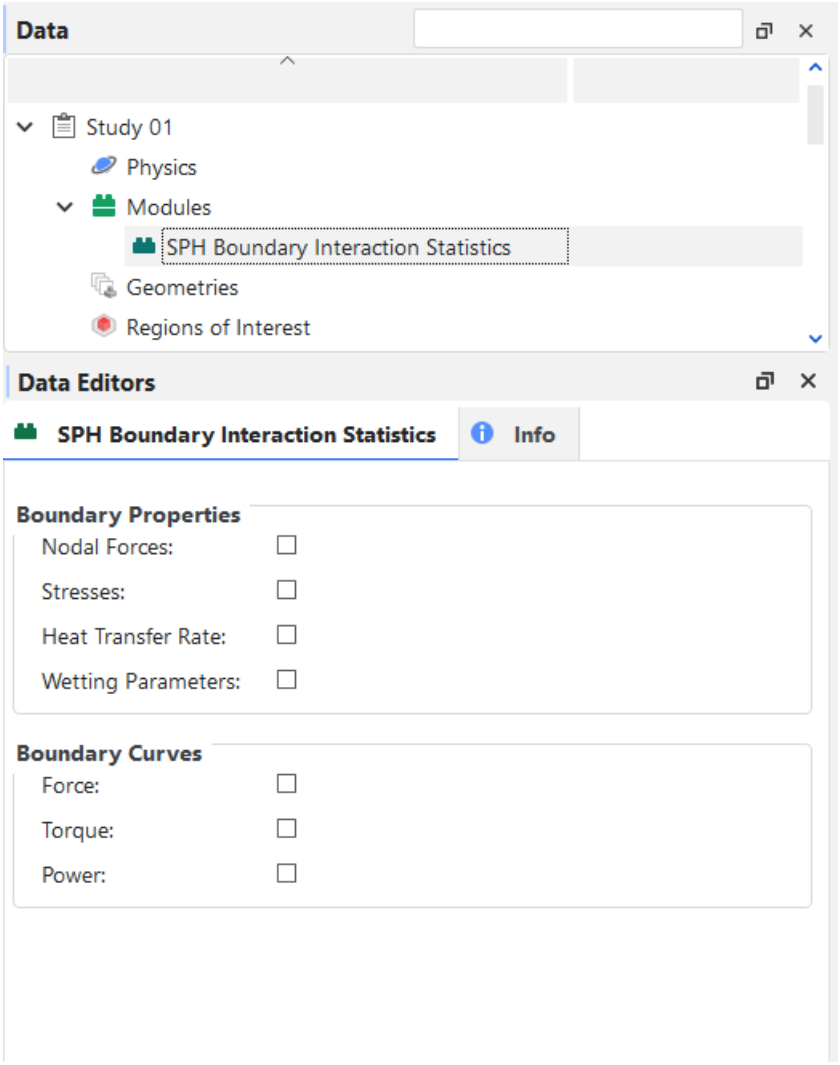
3.7.13. SPH Boundary Interaction Statistics Module

The SPH Boundary Interaction Statistics Module enables a collection of SPH boundary-related data, separated into Boundary Properties, where SPH forces are divided into triangles and the nodal force is collected, and Boundary Curves where the collected value refers to the entire geometry.

MODULE OPTIONS

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

Figure 3.248: Options in the Data Editors panel when the SPH Boundary Interaction Statistics Module is enabled



When the SPH Boundary Interaction Statistics Module is enabled (Figure 1), you are able to select any of the following parameters:

Boundary Properties

- Nodal Forces
- Stresses;
- Heat Transfer Rate;
- Wetting Parameters.

The **Stresses** parameter calculates the time average of the stresses on the triangle. If there is no motion and the Cartesian forces are already stored, stresses are calculated with them, otherwise, normal and tangential components of the forces are stored.

The **Heat transfer rate** parameter calculates the time average of the heat transferred between the fluid and the triangle (positive if the transfer is triangle->fluid, negative otherwise). The **Wetting**

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

parameters calculates the time average of the ratio between the wet area (approximate) and the triangle area.

Note:

Due the use of a kernel function for approximating the wet area, SPH elements located within a distance equal to the kernel radius from a boundary triangle will contribute to that area. This means that if a moving wall approaches an SPH free surface, for instance, it may show wet fractions above zero before actually touching the SPH elements.

Boundary Curves

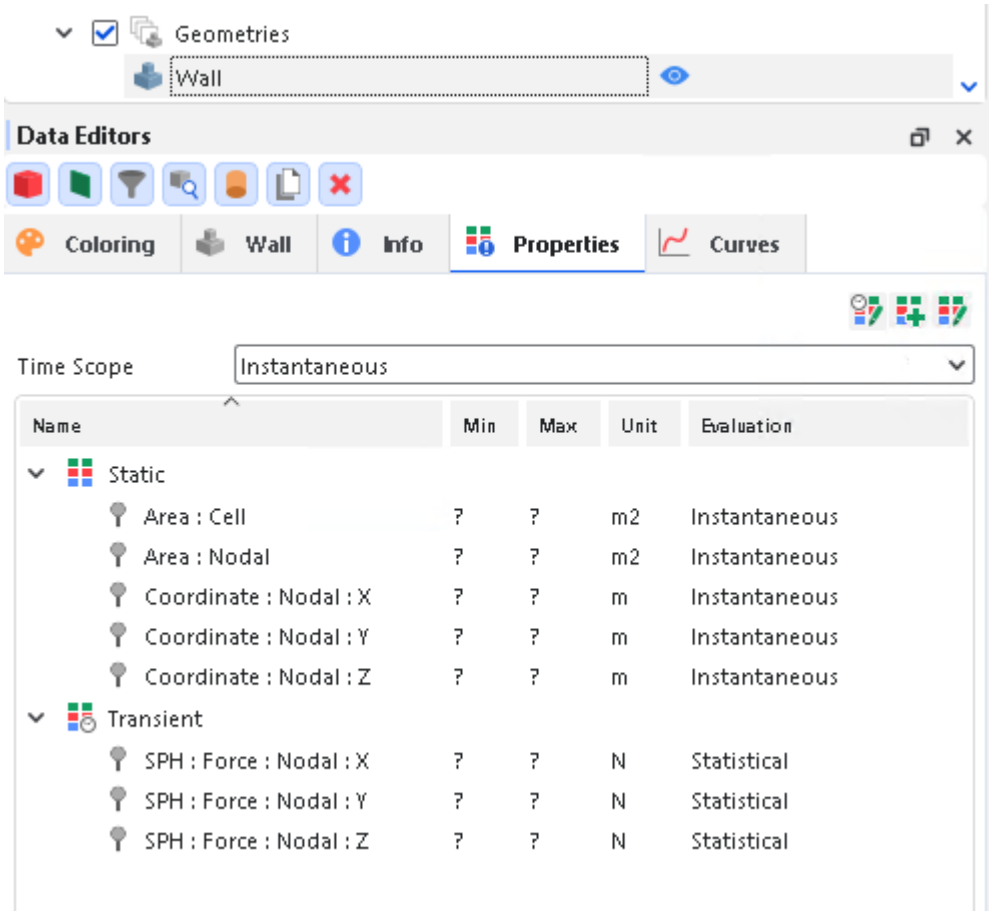
- Force;
- Torque;
- Power.

After processing your simulation, specific data based on the options you enabled will be available in Walls Properties or Curves. As you can see in the figures below:

Walls-Properties

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

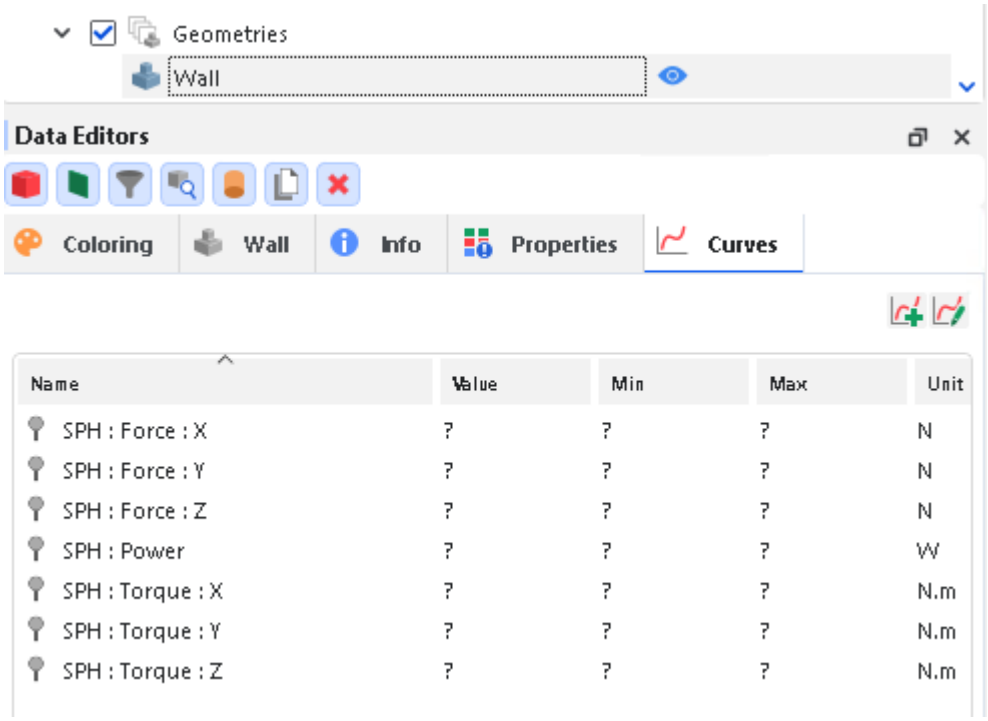
Figure 3.249: Results in Walls Properties when the SPH Boundary Interaction Statistics Module is enabled



Walls-Curves

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

Figure 3.250: Results in Walls Curves when the SPH Boundary Interaction Statistics Module is enabled



What would you like to do next?

- Learn more About Collision Statistics for Boundaries (p. 312)
- About the Boundary Collision Statistics Module (p. 868)
- Learn more About Collecting Data in Rocky (p. 306)

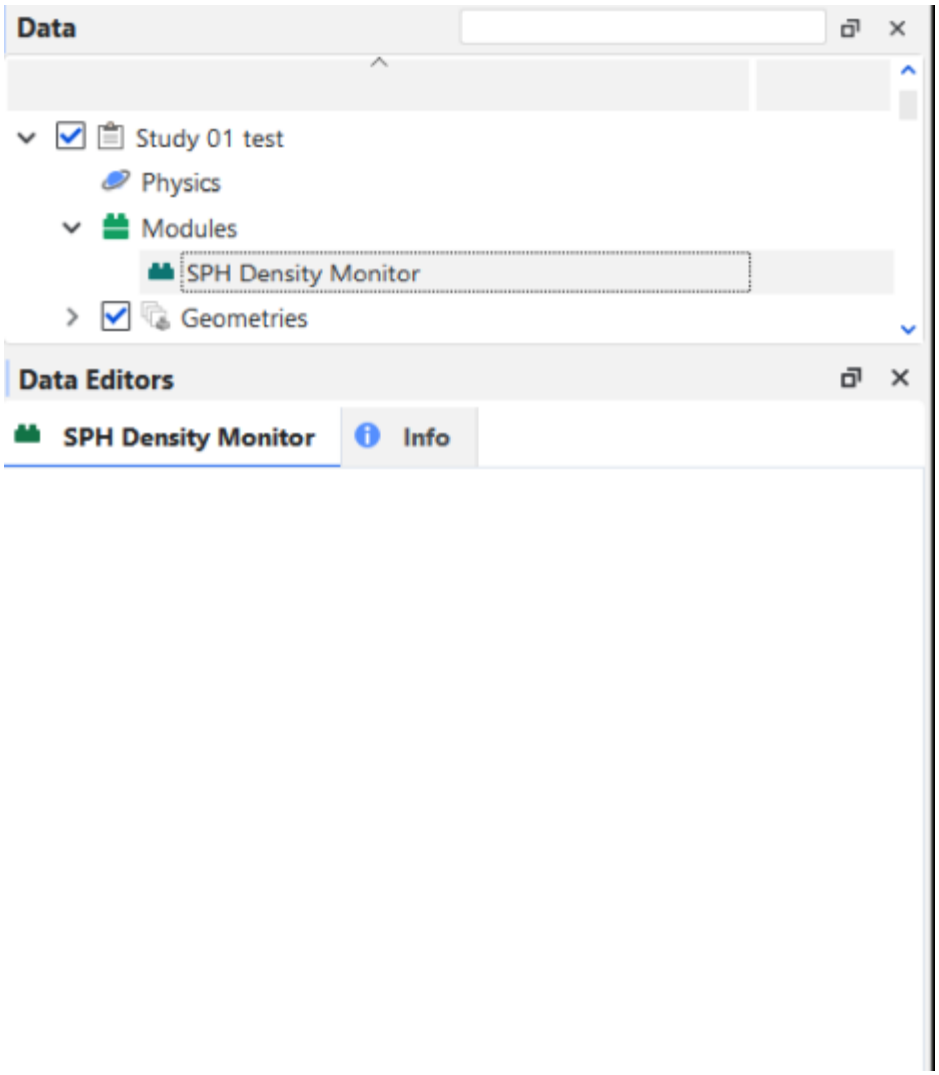
3.7.14. SPH Density Monitor Module

The SPH Density Monitor Module enables the monitoring of density values of the SPH elements during a simulation.

MODULE OPTIONS

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

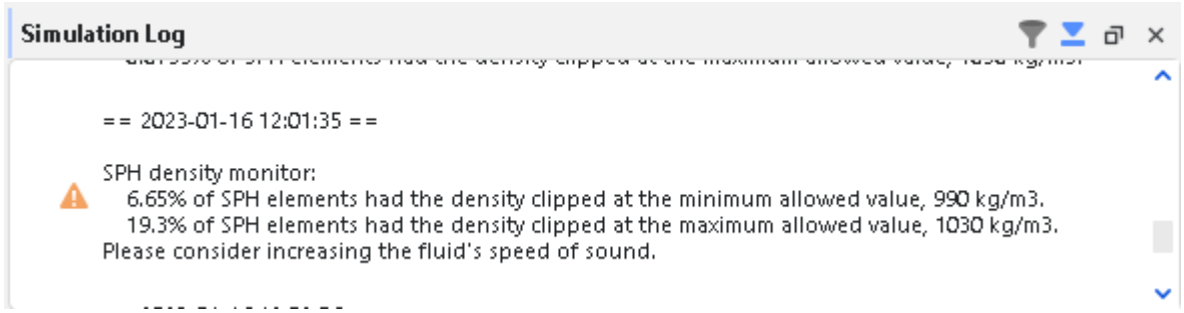
Figure 3.251: Options in the Data Editors panel when the SPH Density Monitor Module is enabled



During the simulation, if the module detects that the density values were clipped due the existence of positive or negative deviations that exceeded the maximum allowed values, it will issue a warning, indicating the percentage of SPH elements that had density values clipped. These Warnings are registered in the log file and can be consulted later.

See the example in the figure below:

Figure 3.252: Simulation Log example when the SPH Density Monitor Module is enabled



Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

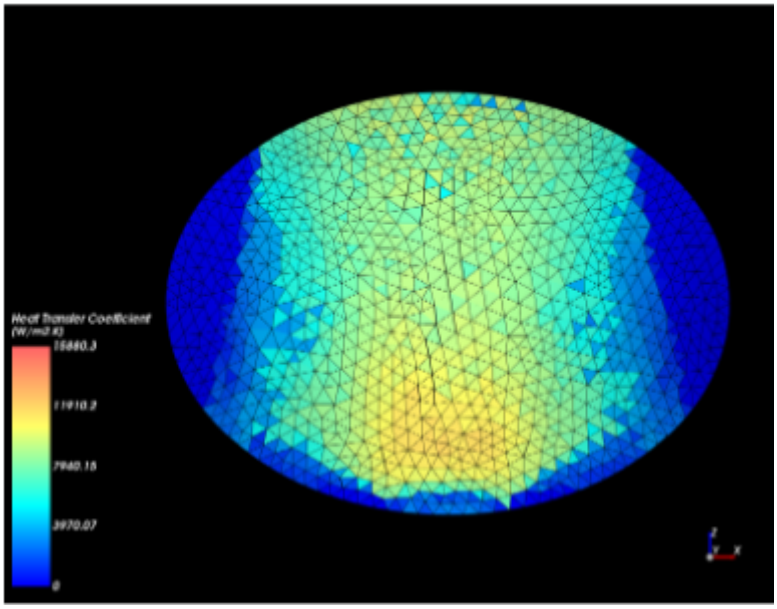
What would you like to do next?

- Learn more Understand Rocky Modules (p. 220)
- Install an External Module (p. 223)
- Learn more About Modules Parameters (p. 431)

3.7.15. SPH HTC Calculator Module

The SPH HTC Calculator module calculates the heat transfer coefficient (HTC) through forced convection for each wall triangle. It provides an estimation of heat exchange without requiring a thermal solution.

Figure 3.253: Example application of heat transfer estimation using the SPH HTC Calculator module.



The heat transfer coefficient is calculated using simple correlations based on local Reynolds and Prandtl numbers. The correlations used for computing the Nusselt number depend on whether the flow is laminar or turbulent, as determined by the user-defined threshold.

Limitations

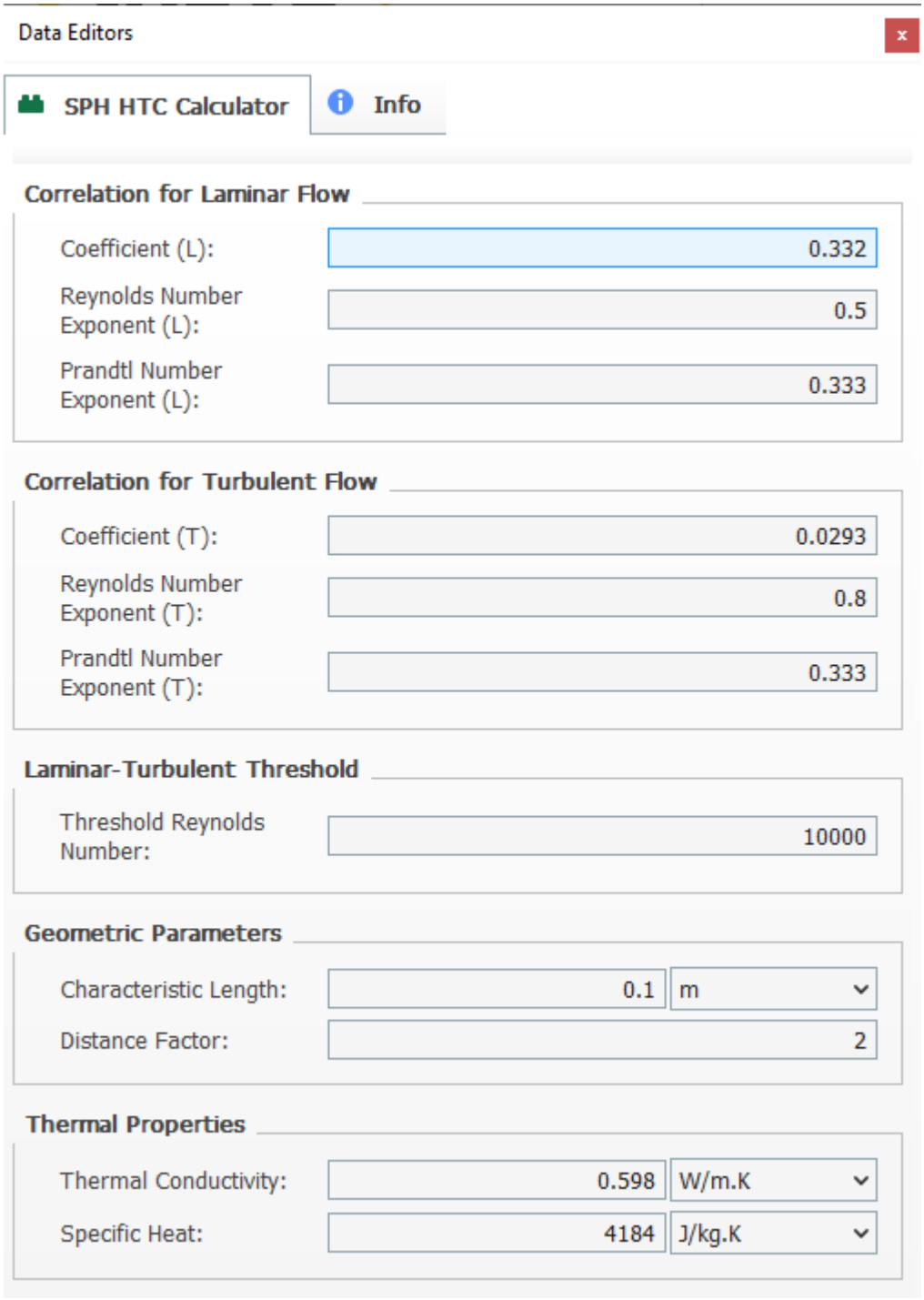
Listed below are known limitations of the SPH HTC Calculator module:

- User input is required for fluid thermal conductivity and specific heat.
- The module ignores the internal values of these properties in the fluid material; however, fluid viscosity is still extracted from the fluid material data.
- The heat transfer coefficient is computed based on the local Nusselt number, which utilizes the fluid velocity at a specific distance from the wall, as determined by the Distance Factor.
- The user has the ability to modify the parameters utilized for calculating the Nusselt number in accordance with the flow characteristics.

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

Module Options

Figure 3.254: SPH HTC Calculator module options.



Refer to Figure 3.254: SPH HTC Calculator module options. (p. 896) and the parameter definitions below to understand how to enable and configure the SPH HTC Calculator module. These parameters are explained below:

- **Coefficient (L):** Coefficient used to compute the Nusselt number for a laminar flow. Range: [Positive values]

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

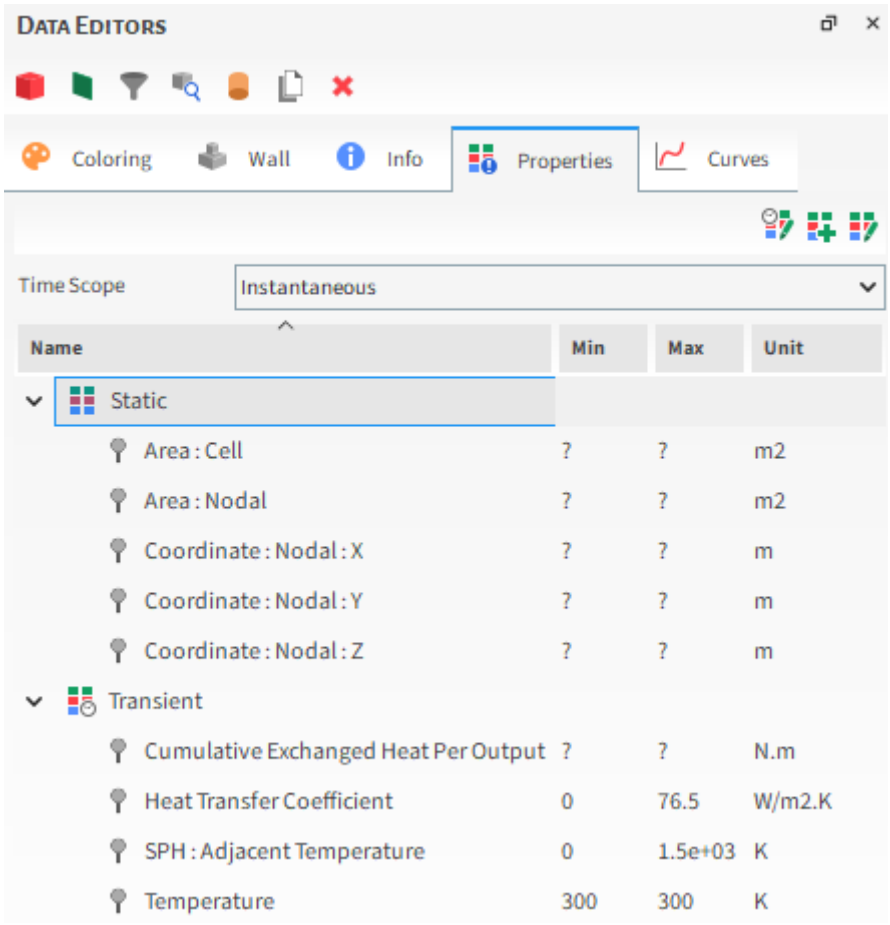
- **Reynolds Number Exponent (L):** Exponent that raises the Reynolds number to compute the Nusselt number for a laminar flow. Range: [Positive values]
- **Prandtl Number Exponent (L):** Exponent that raises the Prandtl number to compute the Nusselt number for a laminar flow. Range: [Positive values]
- **Coefficient (T):** Coefficient used to compute the Nusselt number for a turbulent flow. Range: [Positive values]
- **Reynolds Number Exponent (T):** Exponent that raises the Reynolds number to compute the Nusselt number for a turbulent flow. Range: [Positive values]
- **Prandtl Number Exponent (T):** Exponent that raises the Prandtl number to compute the Nusselt number for a turbulent flow. Range: [Positive values]
- **Threshold Reynolds Number:** This defines the laminar-turbulent transition. Above this Reynolds number, the flow is considered turbulent. Range: [Positive values]
- **Characteristic Length:** Length used to compute the Reynolds number and the heat transfer coefficient. Range: [Positive values]
- **Distance Factor:** Factor that multiplies the SPH element spacing for computing the distance from the wall at which the velocity is obtained. Range: [Positive values]
- **Thermal Conductivity:** This defines the fluid thermal conductivity, which is used to compute the Prandtl number and the heat transfer coefficient. Range: [Positive values]
- **Specific Heat:** This defines the specific heat used to compute the Prandtl number. Range: [Positive values]

Post-processing abilities

After simulating, a new Heat Transfer Coefficient property becomes available for wall entities. This property aids in visualizing heat exchange between the SPH fluid and each wall.

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

Figure 3.255: New wall properties are shown in the Data Editors panel when the SPH HTC Calculator module is enabled.



These new properties are explained below:

Heat Transfer Coefficient: This provides the heat transfer coefficient for each wall triangle.

SPH: Adjacent Temperature This provides the SPH adjacent temperature for each wall triangle. When coupling with mechanical this average adjacent temperature is sent to mechanical.

You can analyze these properties in a plot or histogram window. Alternatively, you can graphically display these properties in a 3D View window.

Setting up and using the module

1. Ensure that the module is enabled. (From the **Data panel**, select **Modules** and then from the **Data Editors panel**, ensure the **SPH HTC Calculator** checkbox is enabled.)
2. From the **Data panel**, under **Modules** , select the new **SPH HTC Calculator** entry.
3. From the **Data Editors panel**, on the **SPH HTC Calculator** tab, enter the values you want.
4. Continue setting up the simulation as you normally would.
5. Process the simulation as you normally would.

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

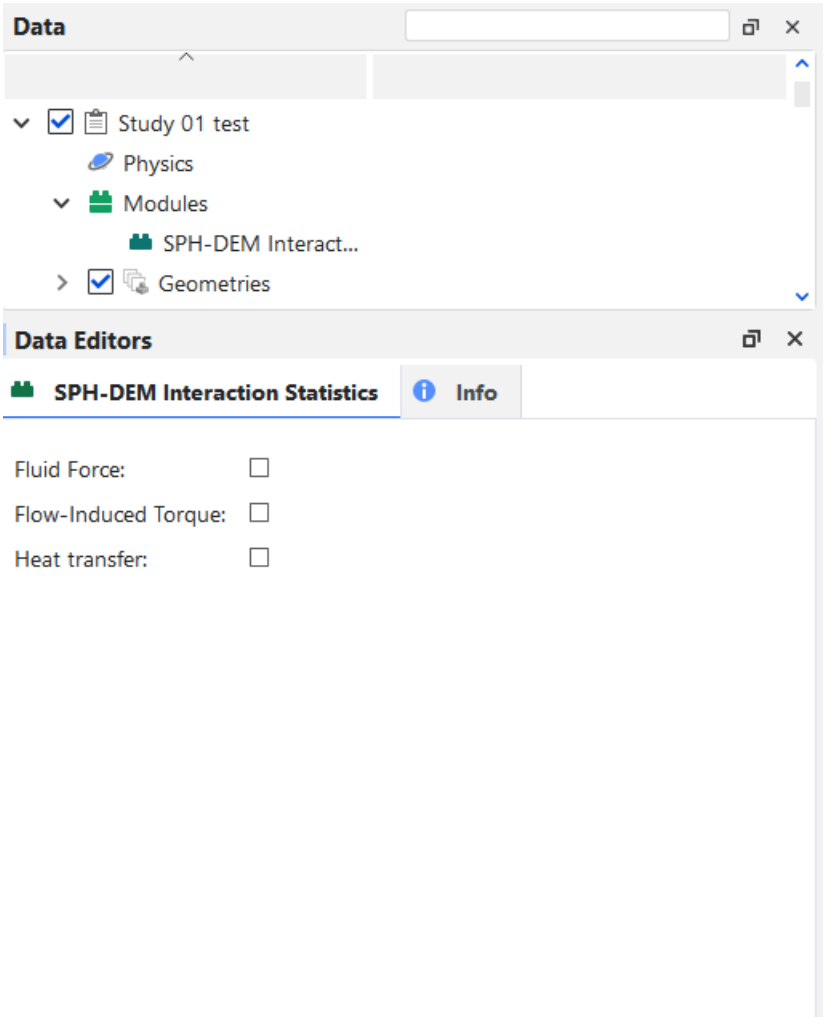
6. When you are ready to post-process your simulation results, you can make use of the new parameter on the **Properties** tab for the main **Particles** entity.

3.7.16. SPH-DEM Interaction Statistics Module

The SPH-DEM Interaction Statistics Module enables the calculation of interaction between particles and fluids within the simulation.

MODULE OPTIONS

Figure 3.256: Options in the Data Editors panel when the SPH DEM Interaction Statistics Module is enabled



When the SPH-DEM Interaction Statistics Module is enabled (Figure 1), you are able to select any of the following Properties:

- Fluid Force
- Flow-Induced torque
- Heat Transfers

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

Important: To Heat Transfers option works, it's necessary that the Enable Thermal checkbox in Physics (p. 417) is selected. Otherwise, an error message will appear.

After processing your simulation, specific SPH Properties will appear along with the Particle (p. 602) parameters for the options you enabled before starting your simulation. You can see some of this properties below:

- SPH:Flow-Induced Torque
- SPH:Flow-Induced Torque: X
- SPH:Flow-Induced Torque: Y
- SPH:Flow-Induced Torque: Z
- SPH:Fluid Force
- SPH:Fluid Force: X
- SPH:Fluid Force: Y
- SPH:Fluid Force: Z
- SPH:Heat Transfer Rate

What would you like to do next?

- Learn more About Adding and Editing Particle Sets (p. 602)
- About the CFD Coupling Particle Statistics Module (p. 873)
- Learn more About SPH Mass Flow Rate Module (p. 887)

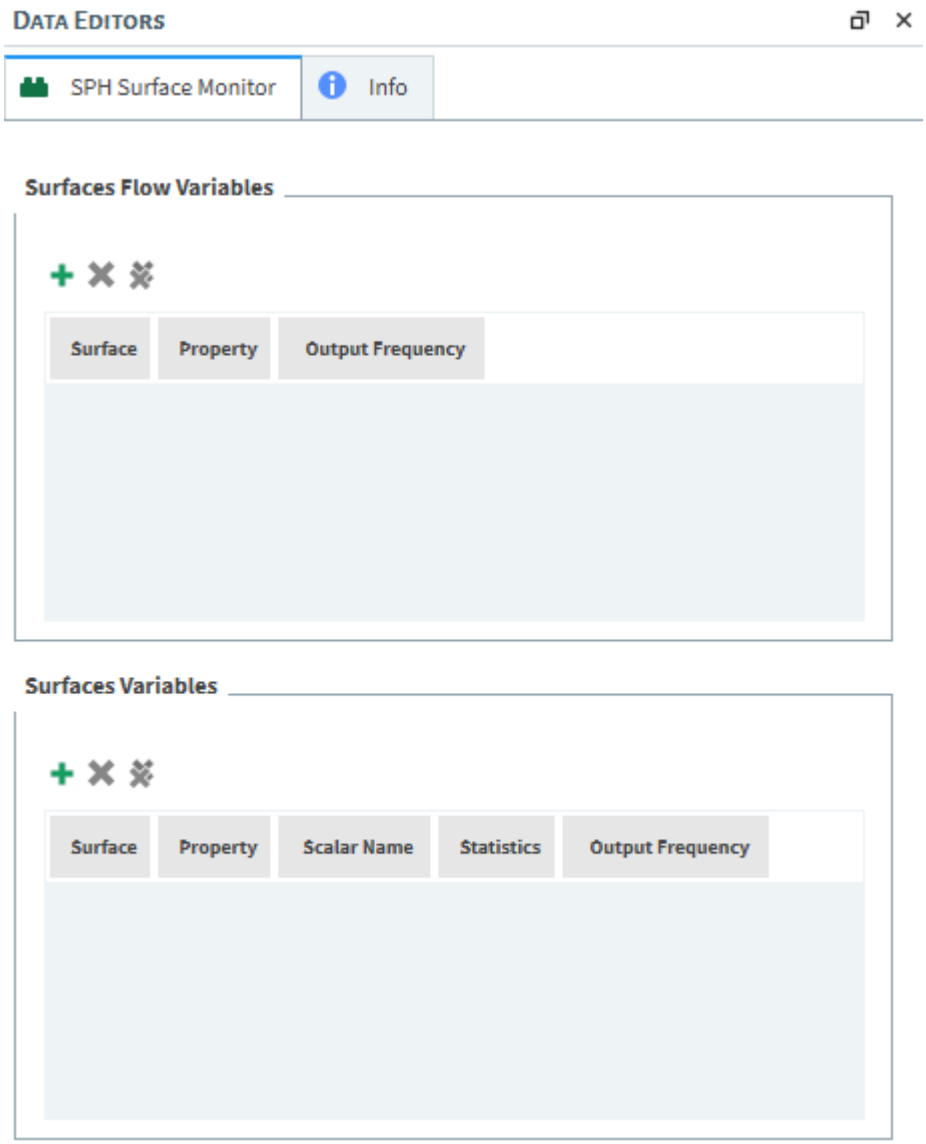
3.7.17. SPH Surface Monitor Module

The SPH Surface Monitor module allows you to select one or more surfaces from the simulation and track SPH properties on those surfaces. Additionally, you can also define the output frequency at which results are provided. This allows you to obtain results at a higher rate than that of the simulation output frequency.

MODULE OPTIONS

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

Figure 3.257: Options in the Data Editors panel when the SPH Surface Monitor Module is enabled



When the SPH Surface Monitor Module is enabled, two fields become available:

- **Surfaces Flow Variables**
- **Surfaces Variables**

The **Surfaces Flow Variables** field allows you to choose between **Mass** and **Count** properties, both at **Output** frequency or **Solver Curves** frequency, for as many surfaces as necessary.

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

Meanwhile, aside from allowing you to select any simulation surface and define the desired output frequency, the **Surfaces Variables** field also enables a wider range of **SPH Properties** and the option to track a **Scalar**. Additionally, it allows you to select different **Statistics** to plot.

Note:

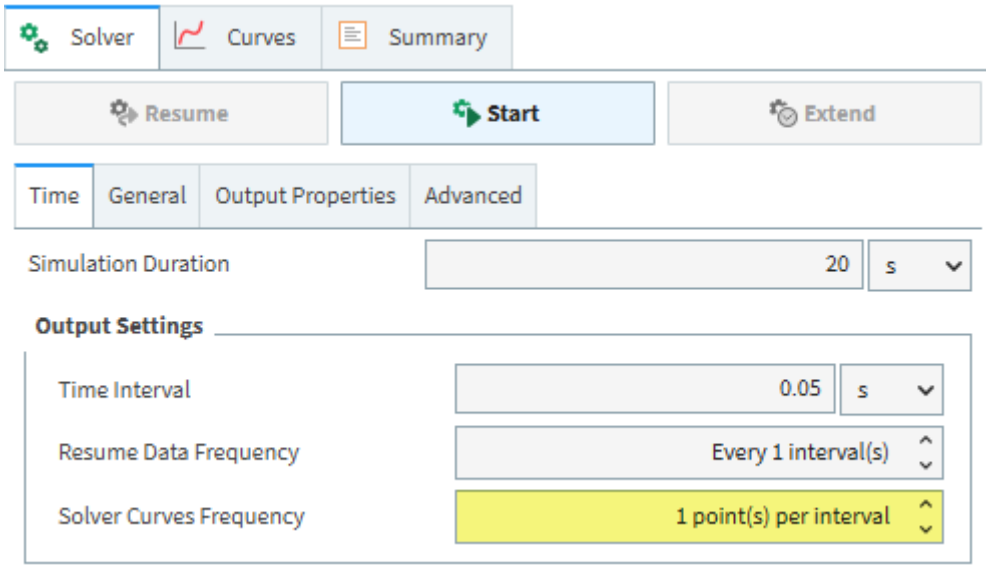
The *Scalar Name* option enables you to analyze a property from another Module in your simulation.

To use the **Surfaces Flow Variables** option, follow the steps below:

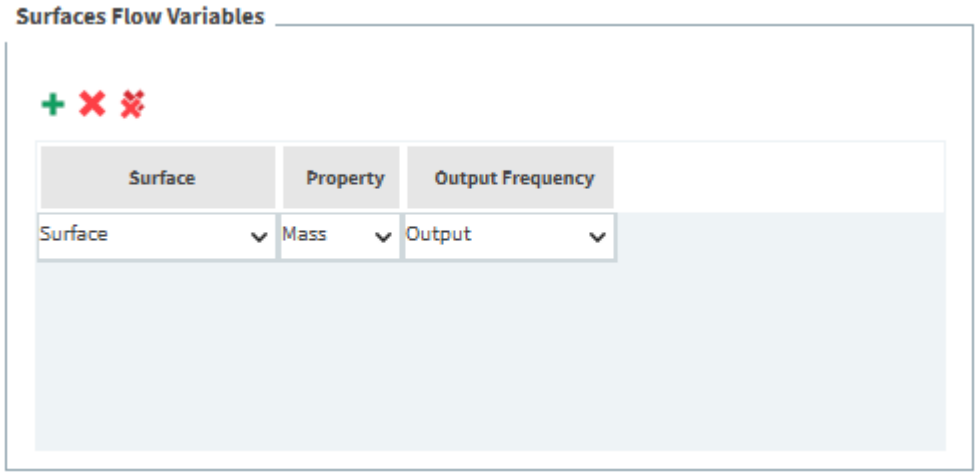
1. In the **Surfaces Flow Variables** field, click the **Add** button to create a new entry.
2. From the **Surface** dropdown list, select the simulation surface you want.
3. From the **Property** dropdown list, select one of the properties:
   - Mass
   - Count
4. From the **Output Frequency** dropdown list, select one of the options:
   - Output
   - Solver Curves
5. If you choose the **Solver Curves** option, there is an additional step to configure the desired frequency. Follow the points below:
   a. From the **Solver** entity, navigate to the **Time** tab.
   b. Under the **Output Settings** field, you will find the **Solver Curves Frequency** entry.

      Here you will define how many times between Time Intervals the Solver will generate outputs for the selected property.

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.



6. See below an example of the Surfaces Flow Variables setup:



To use the **Surfaces Variables** option, follow the steps below:

1. From the **Solver** entity, navigate to the **Time** tab.
2. From the **Surface** dropdown list, select the simulation surface you want.
3. In this option, you have to choose either a **Property** or a **Scalar.**

Important:

Selecting a Property and naming a Scalar in the same entry will cause your simulation to break.

In case you need to analyze an SPH Property, you can choose between:

- Count

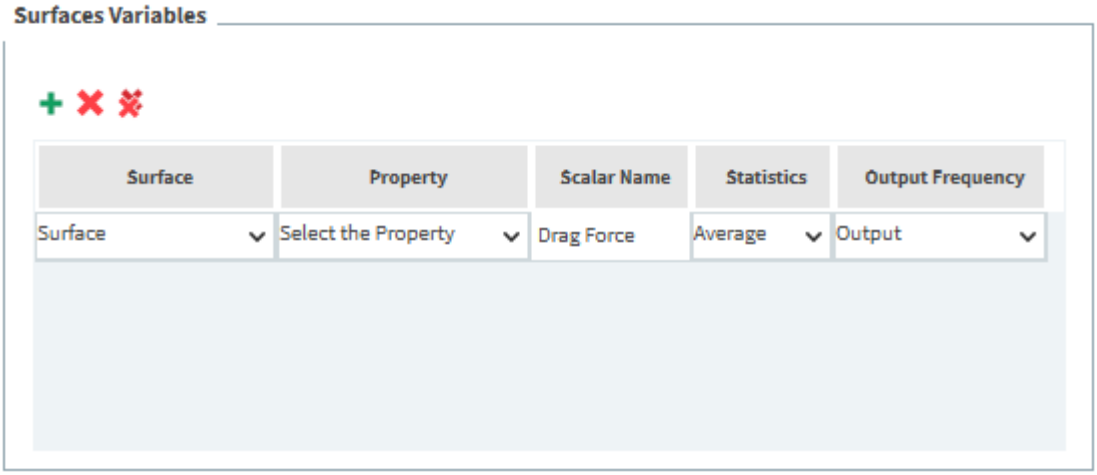
Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

- Density
- Mass
- Pressure
- Temperature
- Velocity
- Turbulent Viscosity
- Molecular Viscosity
- Volume

In case you need to track a Scalar, do the following:

a. This option works if you have another Module enabled, and if said module enables a new property for the SPH entity. In this document, the SPH Air Drag Module will be used as an example. This module enables the *Drag Force* property.

b. In the **Scalar Name** field, type the name of the module's property. As an example, wishing to track the Drag Force provided by the SPH Air Drag Module, you would type "Drag Force".



4. Next, from the **Statistics** dropdown list, select the statistic you wish:
   - Average
   - Minimum
   - Maximum
   - Sum
5. From the **Output Frequency** dropdown list, select one of the options:
   - Output

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

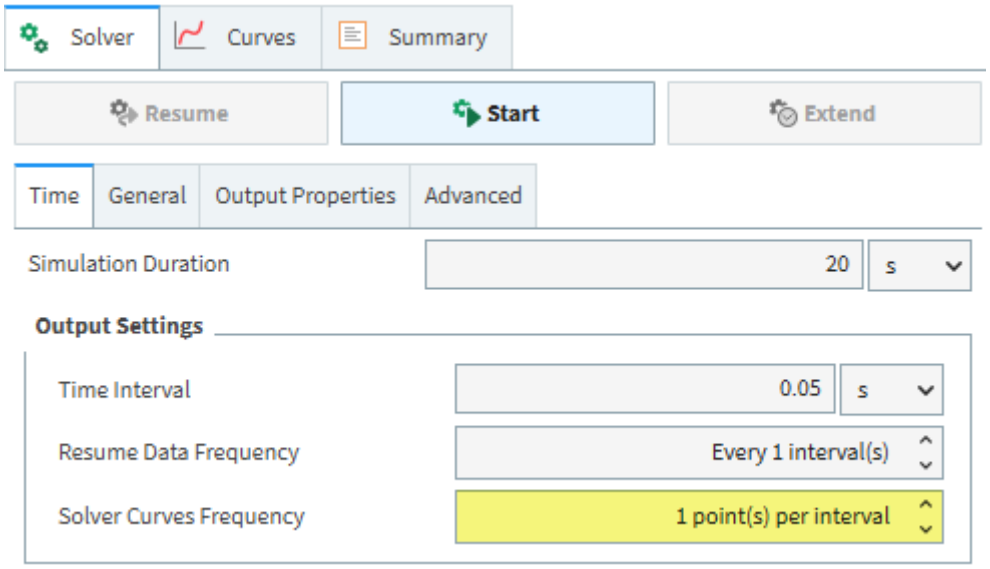
- Solver Curves

6. If you choose the **Solver Curves** option, there is an additional step to configure the desired frequency. Follow the points below:

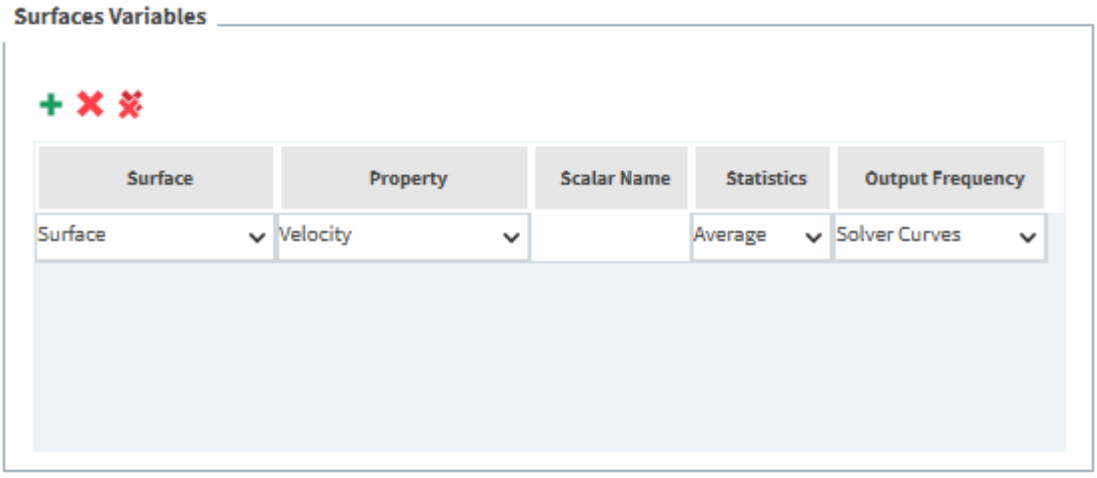
   a. From the **Solver** entity, navigate to the **Time** tab.

   b. Under the **Output Settings** field, you will find the **Solver Curves Frequency** entry.

      Here you will define how many times between Time Intervals the Solver will generate outputs for the selected property.



7. See below an example of the Surfaces Variables setup:



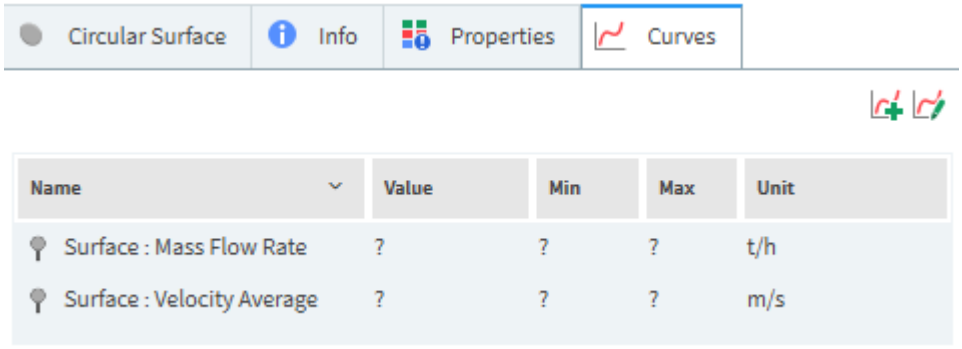
Note:

You can always create a new entry to cover all of the desired surfaces, properties and statistics.

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

When the setup is finished, you can run the simulation.

After the simulation has been processed, the resulting curves will be available at the **Curves** tab of the surfaces that you selected as entries in the SPH Surface Monitor Module.



Important:

- You can make use of this module to track a certain simulation property until it reaches a steady-state, for example, so it will be up to you whether to continue or stop the simulation.
- In some cases, there are only a few properties of interest, you can use this module to evaluate the desired properties and disable the collection of those that are not important in the context of your simulation, gaining processing speed and generating less data.

What would you like to do next?

- About the CFD Coupling Particle Statistics Module (p. 873)
- Learn more About SPH Mass Flow Rate Module (p. 887)

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

Chapter 4: Processing a Simulation

After you have set up your simulation (see Setting Up a Simulation (p. 411)), you are ready to process it. Processing is the step in which Rocky calculates at each progressive time step (p. 1395) how each individual particle interacts with every other particle and boundary it comes into contact with.

The time it takes for your simulation to complete will depend upon the complexity of the parameters you have set, and the number and type of the processors you have chosen. Based upon these characteristics, your simulation can take anywhere from a few minutes to several days to complete. (See also Best Practices for Faster Processing (p. 68).)

Important:

You are only able to set and change setup parameters prior to processing your simulation so ensure that your settings are correct before you begin processing. (See also I cannot change my setup parameters during processing (p. 1361).)

While your simulation is processing, you might stop and restart the simulation several times before your simulation is complete. As the simulation is processing, you can see the progress in a 3D View window or see data collect in a plot or histogram. You can also set multiple simulations to be processed in batch-like succession by using the Rocky Scheduler, or even run the simulation from the command line without even opening Rocky.

What would you like learn more about?

- Starting a Simulation (p. 907)
- Stopping a Simulation (p. 914)
- Extending the Duration of an In-Process or Completed Simulation (p. 915)
- Processing Multiple Simulations In Succession (p. 919)
- Use the Command Line to Process a Simulation (p. 1311)

4.1. Starting a Simulation

In Rocky, starting a simulation can involve any of the following three scenarios:

- You have finished setting your simulation parameters and are ready to start processing it.
- You have stopped a simulation that was already processing and want to resume processing it from where it left off.
- You have stopped a simulation that was already processing and want to start the processing over from the very beginning.

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

What would you like to do?

- Learn more About Starting a Simulation (p. 908)
- Start Processing a Simulation from the Beginning (p. 913)
- Resume Processing a Stopped Simulation (p. 913)

See Also:

- Processing a Simulation (p. 907)

4.1.1. About Starting a Simulation

When you start processing a simulation, you select whether you want CPU, GPU, or Multi GPU processing. If CPU processing is chosen, then you are asked to choose the number of processors. If GPU is selected, then you choose which one graphics card you want to use. If Multi GPU is selected, you choose which two or more graphics cards you want to use, and what slicing direction you want processing to occur.

Note:

Even when you select GPU or Multi GPU processing for your simulation, Rocky still requires the use of at least one CPU core to complete certain pre-processing tasks. These tasks include calculating what particles will be released into the simulation (all Inputs) and for Volume Fill Inputs specifically, generating and positioning the particles prior to processing. After these pre-processing tasks are complete, Rocky can use either CPU, GPU, or Multi GPU to process the simulation.

When any amount of GPU is selected, you also have the option of viewing details of the GPU card(s) you selected by clicking the **GPU Information** button, which will display details in a separate dialog (Figure 1).

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

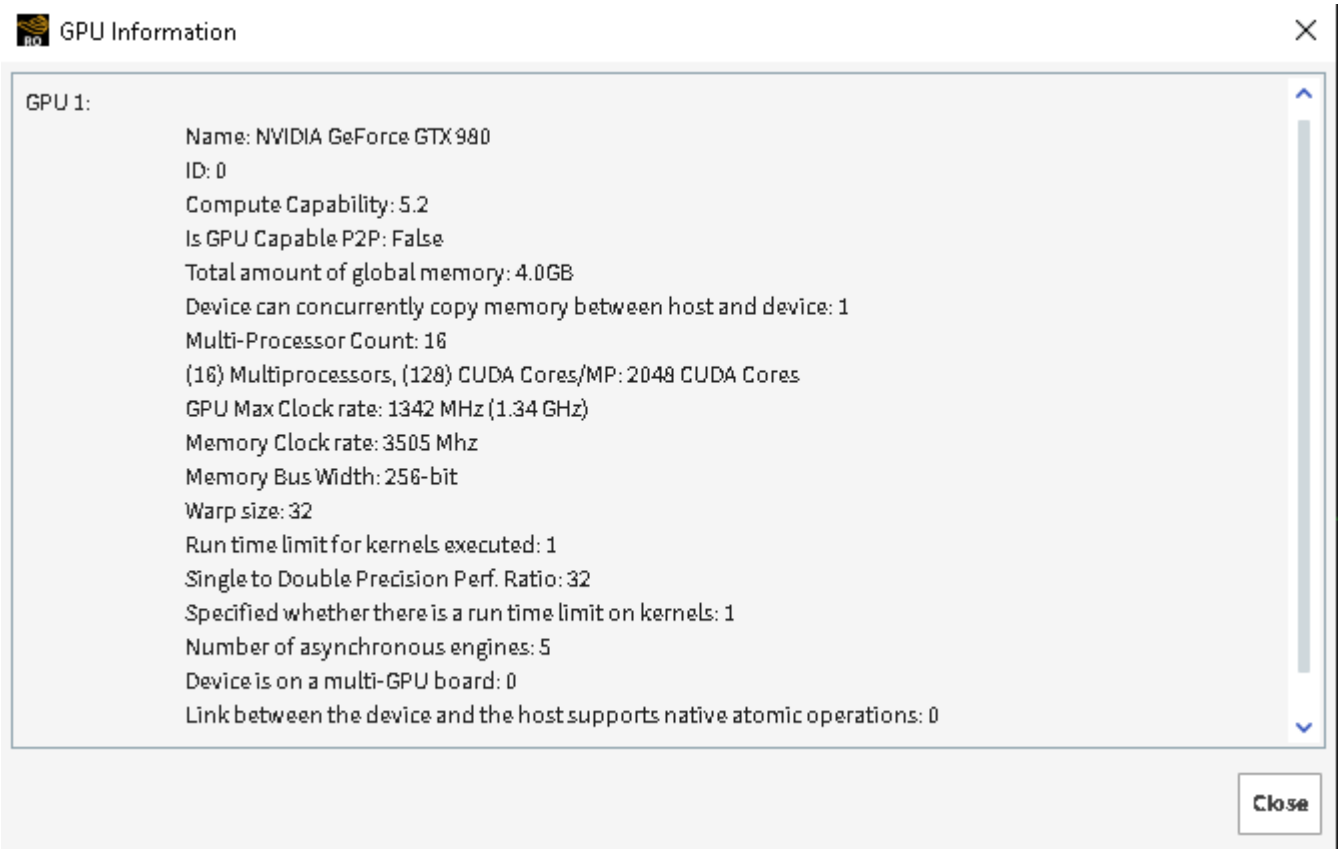


Figure 1: GPU Information dialog

While the simulation is processing, you are able to use Rocky to review your simulation settings, create charts, graphs, and 3D Views of the data collected thus far, and more.

The only task you are unable to do while a simulation is processing is change the set up parameters. Doing this would affect the calculations that Rocky is solving. To change the setup parameters, first stop the simulation, change your parameters, and then start your simulation over from the beginning.

In addition to the information in the Rocky Title Bar (see also: Refer to the Rocky Title Bar for Simulation Progress Details (p. 119)), a separate progress bar is provided to help you track the progress of your simulation, and provides Stop and Refresh capabilities (Figure 2).

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

Figure 2: Rocky simulation progress options during processing

Because processing is so resource intensive, Rocky does not automatically update the 3D Views and graphs windows with the new information it calculates. Rather, you can choose to refresh your views manually at any time, or can even choose to override the default setting and have the views refreshed automatically at each time step.

Important When choosing to processes very long or complex cases over a network, it is important to periodically check the status of your simulation to ensure that loss of connectivity or other issues haven't stopped it from processing. (See also I get a "Failed to write the log" message while processing my simulation (p. 1340).)

Note:

As of Rocky v4.3.0, all simulation files will be made smaller during processing through compression, which takes up less storage on your computer. Because choosing to process your simulation without compression is no longer possible, the option **Use Compressed Files** was removed.

Solver Execution Parameter Definitions

Use the images and table below to help you understand how to start processing your simulation.

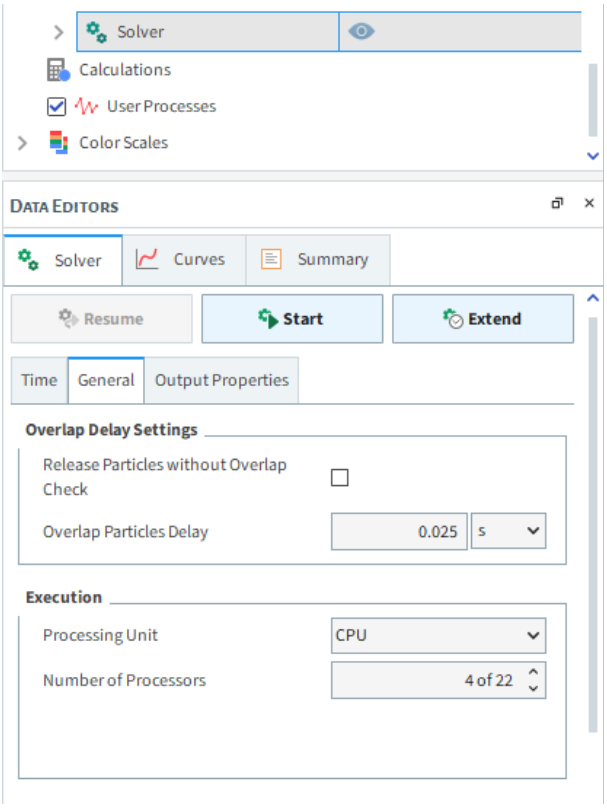


Figure 3: Solver, General parameters in the Data Editors panel when CPU is selected

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

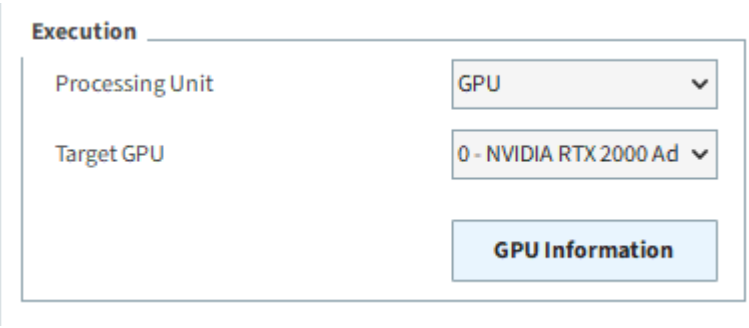


Figure 4: Solver, General, Execution parameters when GPU is selected

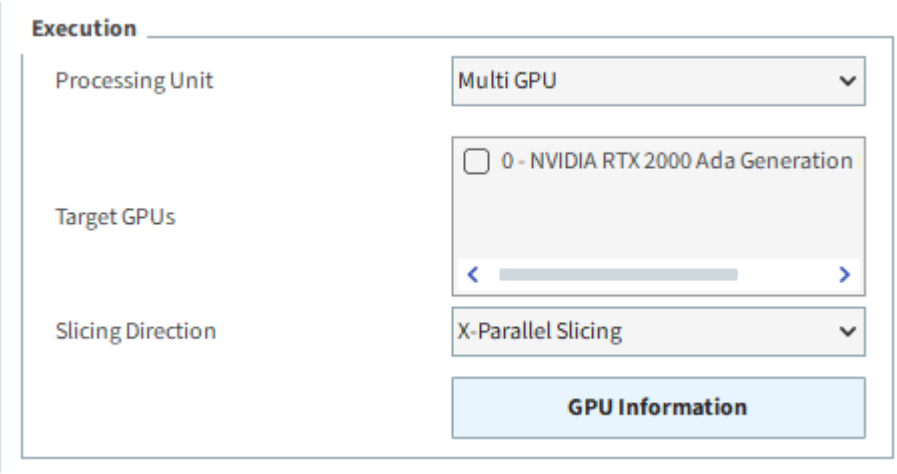


Figure 5: Solver, General, Execution parameters when Multi GPU is selected

Table 1: Solver, General tab, Execution options

Setting	Description	Range
Processing Unit	Enables you to select what portion of your computer will be doing the processing. Notes: • Even when you select GPU or Multi GPU, Rocky will still use at least 1 CPU core to complete certain pre- processing tasks related to Particle Inputs. After these tasks are complete, Rocky will process the simulation using only the resources you select here. • In this version of Rocky, Multi GPU processing is incompatible with the Radl et al. Coarse Grain Model. (See also Set Up a New GPU Card (p. 404).)	CPU; GPU; Multi GPU
Number of Processors	When CPU is selected, sets the number of computer processors (threads) the simulation will use. In general, the more processors used, the faster the simulation will complete.	Positive values

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

Setting	Description	Range
	It is recommended that you set this number equal to the amount of available cores (both physical and virtual) for which you have Rocky licenses. Notes: • Setting more processors than what you have available might slow the simulation. • Windows limits the number of processors to 64 so if you are running Rocky on Windows, it is best to set this value to 64 or less. • Linux does not limit the number of processors you can set.	
Detected Number of Processors	Displays the number of virtual and physical processors that Rocky was able to locate on the current machine. Using this value can help you determine what best to set for **Number of Processors**.	Display only
Target GPU(s)	When GPU or Multi GPU are selected, enables you to select which graphics card(s) will be doing the processing. **Note:** In this version of Rocky, there is no limit to how many GPUs you can use at one time.	Automatically determined
Slicing Direction	When **Multi GPU** is selected for **Target GPUs**, this sets which direction parallel slicing is made during processing. In general, choose the direction that represents the longest boundary component in your simulation. For example, if your longest boundary is a conveyor oriented along the X-axis, choose X-Parallel Slicing. This setting can help reduce processing time.	X-Parallel Slicing; Y-Parallel Slicing; Z-Parallel Slicing

What would you like to do?

- Start Processing a Simulation from the Beginning (p. 913)
- Resume Processing a Stopped Simulation (p. 913)
- Set Up a New GPU Card (p. 404)

See Also:

- Processing a Simulation (p. 907)
- Display and Processing Limitations (p. 66)

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

4.1.2. Start Processing a Simulation from the Beginning

1. From the **Data** panel, click **Solver** and then from the **Data Editors** panel, ensure the **Solver | General** tab is active.
2. Enter what you want for the **Processing Unit**, then do one of the following:
   - If you chose CPU, then enter what you want for the **Number of Processors**.
   - If you chose GPU, then select the one GPU you want from the **Target GPU** list. Note: To see detailed information about the GPU you selected, click the **GPU Information** button.
   - If you chose Multi GPUs, then select one or more GPUs you want from the **Target GPUs** list, and indicate the **Slicing Direction** you want. Note: To see detailed information about the GPUs you selected, click the **GPU Information** button.
3. Click **Start**. The Simulation Summary dialog appears, which you can review while Rocky initializes the simulation. (See also About the Simulation Summary (p. 859).) After initialization is complete, Rocky automatically closes the Simulation Summary dialog and begins processing the simulation. The simulation progress bar appears at the bottom of the screen and processing begins at the zero time step. (See also Refer to the Rocky Title Bar for Simulation Progress Details (p. 119).) When processing is complete, the progress bar disappears.

Important When choosing to processes very long or complex cases over a network, it is important to periodically check the status of your simulation to ensure that loss of connectivity or other issues haven't stopped it from processing. (See also I get a "Failed to write the log" message while processing my simulation (p. 1340).)

Tips:

- To see the progress of your simulation in a 3D View window, on the progress bar at the bottom of your Rocky screen, click **Refresh** and then use the **Play Simulation** button on the Time toolbar. (See also About the Time Toolbar (p. 1279).)
- To always see the latest results, select the **Auto Refresh** check box on the progress bar. Note: Selecting this option will take up more processing power than choosing to **Refresh** when you want on a manual basis.

See Also:

- About Starting a Simulation (p. 908)
- Resume Processing a Stopped Simulation (p. 913)

4.1.3. Resume Processing a Stopped Simulation

- From the **Solver** tab, click **Resume**. Processing continues from the last time step that was saved.

Notes:

- Whether or not you have enabled the **Collect Contacts Data** checkbox on the **Contacts** entity will determine how quickly your simulation resumes processing. See About Contacts (p. 746) for details.

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

- You may only resume processing your stopped simulation if the simulation was last processed in the same major version of Rocky as the version you are using now. (See also I cannot resume processing my older simulation in the latest version of Rocky (p. 1359).)

See Also:

- About Starting a Simulation (p. 908)
- Start Processing a Simulation from the Beginning (p. 913)

4.1.4. Starting a Simulation with MPI (Message Passing Interface)

The MPI implementation is a feature available since Rocky 2026 R1. In 2026 R1, the support is only for spherical particle shape, with no support for coupling, no support for external modules or built-in modules in which inter-node reductions are required.

Two levels of parallelism are implemented in Rocky, threads (OpenMP) and processes (Message Passing Interface, a.k.a, MPI). Besides that, MPI has support for distributed parallel computing, i.e., MPI enables Rocky solver to run in distributed memory architetures ("inter-node"). Once with distributed parallel computing support, Rocky users can run simulations in multiple computers (or nodes) connected by a network, such as, InfiniBand. Hence, there will be an improvement on performance scalability for DEM sphere simulations. However, the GPU support is only available for Linux systems.

To use MPI, one should use `--use-mpi` when calling Rocky. Moreover, alongside this flag at least one other flag should be used to specify the amount of computational resources for Rocky solver. All available flags are shown in MPI Flags (p. 1309).

On Windows systems, you can run CPU, multi-CPU jobs on diferent nodes (computers) using MPI. Regarding GPU, the MPI on Rocky does not have support for Windows. On Linux systems, CPU and GPU are fully supported. Then, you can run CPU or GPU jobs in multi-node hardwares.

4.2. Stopping a Simulation

There are several reasons why you might want to stop your simulation from processing. For example:

- When you want your computer to take a break from processing, perhaps to do some computer maintenance, with the intent of resuming your simulation again later.
- When you want to prevent Rocky from making any further calculations to the simulation, perhaps because you want to change the simulation parameters and start processing again from the beginning.
- When you have determined you have enough data from the simulation results and no longer need further calculations to be performed.

What would you like to do?

- Learn more About Stopping a Simulation (p. 915)
- Stop a Simulation from Processing (p. 915)

See Also:

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

- Processing a Simulation (p. 907)

4.2.1. About Stopping a Simulation

When you stop a simulation, you can choose to resume it at any time to continue processing from the last saved time step. If you decide to change any of the setup parameters, however, you must start processing your simulation from the beginning.

What would you like to do?

- Stop a Simulation from Processing (p. 915)

See Also:

- About Starting a Simulation (p. 908)
- Processing a Simulation (p. 907)

4.2.2. Stop a Simulation from Processing

- From the **Solver** tab, click **Stop**. Tip:You can also click the **Stop** button on the progress bar at the bottom of your Rocky screen.

See Also:

- About Stopping a Simulation (p. 915)
- Processing a Simulation (p. 907)

4.3. Extending the Duration of an In-Process or Completed Simulation

If you have already started processing your simulation and discover that your Duration value was not set long enough to meet your needs, you can add more time by extending the simulation.

What would you like to do?

- Learn more About Extending the Duration of an In-Process or Completed Simulation (p. 915)
- Extend the Duration of an In-Process or Completed Simulation (p. 919)

See Also:

- Processing a Simulation (p. 907)

4.3.1. About Extending the Duration of an In-Process or Completed Simulation

If you have already started processing a simulation, no matter whether it is still in-process or has completed, extending the duration will add more seconds to the existing Duration value, and will cause the number of particles within the simulation to be recalculated accordingly.

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

Extending the duration also allows you to extend the particle release time for any inlets within the simulation, as well as extend any motions within the Motion Frames that you have defined.

Once you have extended the duration of your simulation, you must then Resume Processing the Simulation (p. 913) in order for processing to continue to the newly defined end point.

Tip:

To see a walk-through example of extending the duration of your simulation, refer to the following Workshop: Tutorial - Vibrating Screen in the *Rocky Tutorial Guide*

Use the figure and table below to understand the various properties you can set when extending the duration of your simulation.

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

Figure 4.1: Extend Simulation dialog

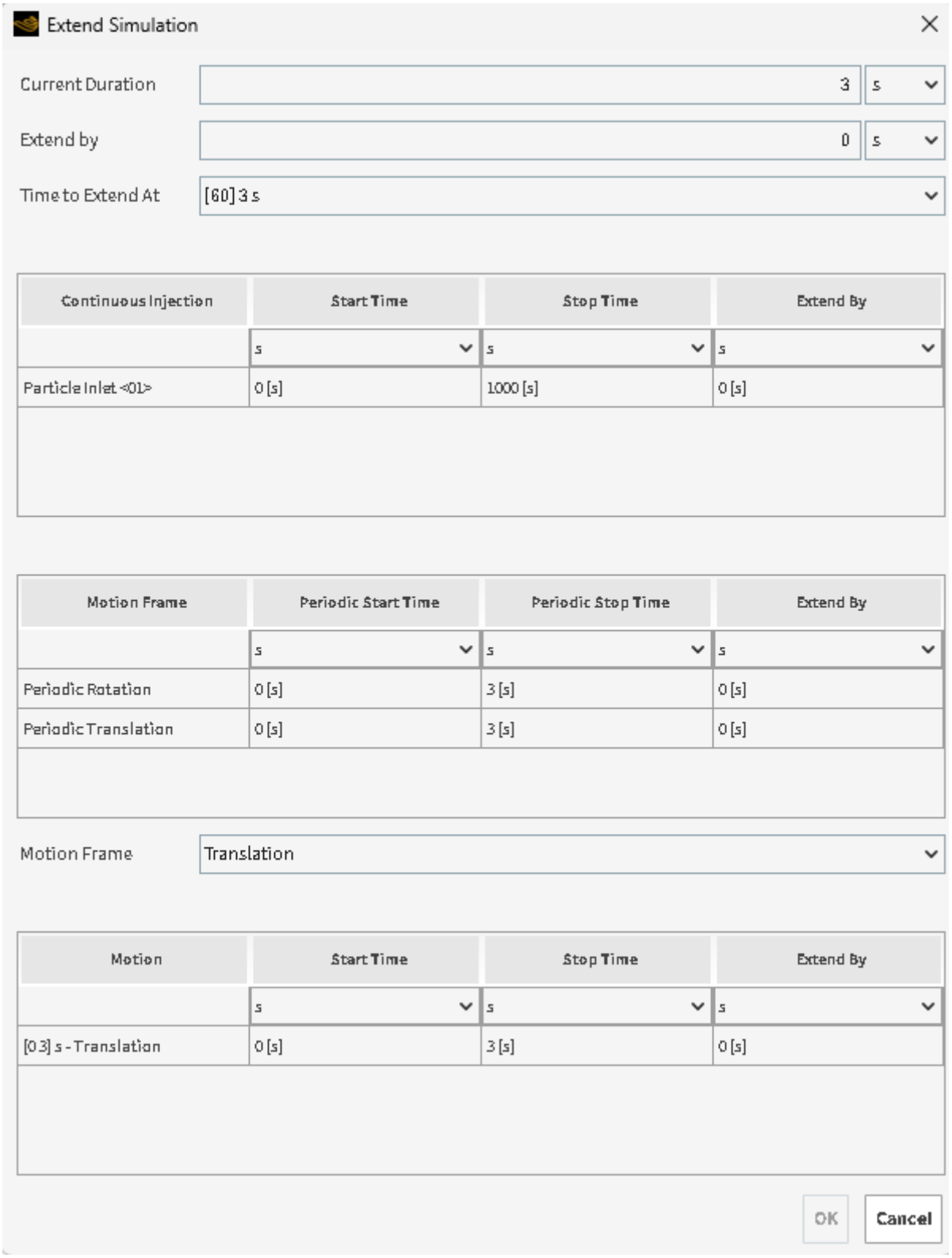


Table 1 : Extend Simulation dialog options

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

Setting	Description	Range
Current Duration	Displays what was originally entered in the **Simulation Duration** text box.	Display only
Extend by	Enables you to choose how much more simulation time you want to add onto your original **Simulation Duration** value.	Positive values
Time to Extend At:	Determines the time step at which the simulation will continue processing when **Resume** is clicked on the **Solver** tab. By default, this displays the last saved time step that has been processed. If you choose an earlier time step from the list, then the time steps that follow will be re-calculated when processing resumes. Note: Changing this from the default should be necessary only if you have inlets, motions or periodic motions you want to extend the duration of that also have original **Stop Times** that end before the current time step. In these cases, to have inlets, motions or periodic motions re-calculated to include the extended time, you must change the **Time to Extend At** value to be less than the inlet's, motion's or periodic motion's **Stop Time** value.	Automatically determined
Inlet	If you have any inlets in your simulation, the names of each inlet are listed here in separate rows.	Display only
Start Time	Displays what was originally entered in the Start Time text box for the inlet or motion indicated.	Display only
Stop Time	Displays what was originally entered in the Stop Time text box for the inlet or motion indicated.	Display only
Motion Frame	Lists in separate rows the names of each periodic motion that is defined in the simulation.	Automatically determined
Periodic Start Time	Displays what was originally entered in the Periodic Start Time text box for the periodic motion indicated.	Display only
Periodic Stop Time	Displays what was originally entered in the Periodic Stop Time text box for the periodic motion indicated.	Display only
Extend by	For an inlet, enables you to extend the amount of time by which particles are released from the inlet. For a motion or periodic motion, enables you to extend the amount of time the motion should occur. **Note:** In order to extend an inlet's, motion's or periodic motion's time, the **Time to Extend At** value must be less than the inlet's or motion's **Stop Time** value.	Positive values
Motion Frame	Enables you to select which Motion Frame contains the motions that you want to extend.	Automatically determined

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

Setting	Description	Range
(drop-down list)		
Motion	Lists in separate rows the names of each motion that is defined in the selected **Motion Frame**.	Automatically determined

What do you want to do?

- Extend the Duration of an In-Process or Completed Simulation (p. 919)

See Also:

- Processing a Simulation (p. 907)
- About Inlet Parameters
- About Creating and Applying Motion Frames (p. 533)

4.3.2. Extend the Duration of an In-Process or Completed Simulation

1. From your stopped but in-process or completed simulation, on the **Solver** tab, click **Extend**.

   Note:

   If **Extend** is disabled, you might need to stop your simulation first. (See also Stopping a Simulation (p. 914).)

2. From the Extend Simulation dialog, choose the values you want, and then click **OK**. The **Simulation Duration** value on the **Time** tab is extended by the amount you chose.

See Also:

- About Extending the Duration of an In-Process or Completed Simulation (p. 915)
- Processing a Simulation (p. 907)

4.4. Processing Multiple Simulations In Succession

If you have many simulations to process on a particular computer or server, you can use the Rocky Scheduler to set up batch processing.

What would you like to do?

- Learn more About Processing Multiple Simulations In Succession (p. 920)
- Open the Rocky Scheduler Program (p. 927)
- Add a New Resource to the Rocky Scheduler (p. 928)
- Remove a Resource from the Rocky Scheduler (p. 928)

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

- Process Multiple Simulation Projects In Succession (p. 929)

See Also:

- Processing a Simulation (p. 907)

4.4.1. About Processing Multiple Simulations In Succession

Rocky Scheduler is a separate program that enables you to have multiple simulation projects set to process sequentially even when you are not physically at your computer to start them. Once one simulation completes processing, the next one in the list begins processing, and so on.

The Rocky Scheduler uses resources you set to specify the hardware, for example, your CPU or GPU(s), that you want to use for processing. In order to process a job using the Rocky Scheduler, you must have at least one resource assigned to it. Each resource used will require one instance of the Rocky solver to be used for processing. So even though you can set as many resources as you like, you are limited by your Rocky license as to how many jobs you can process at once.

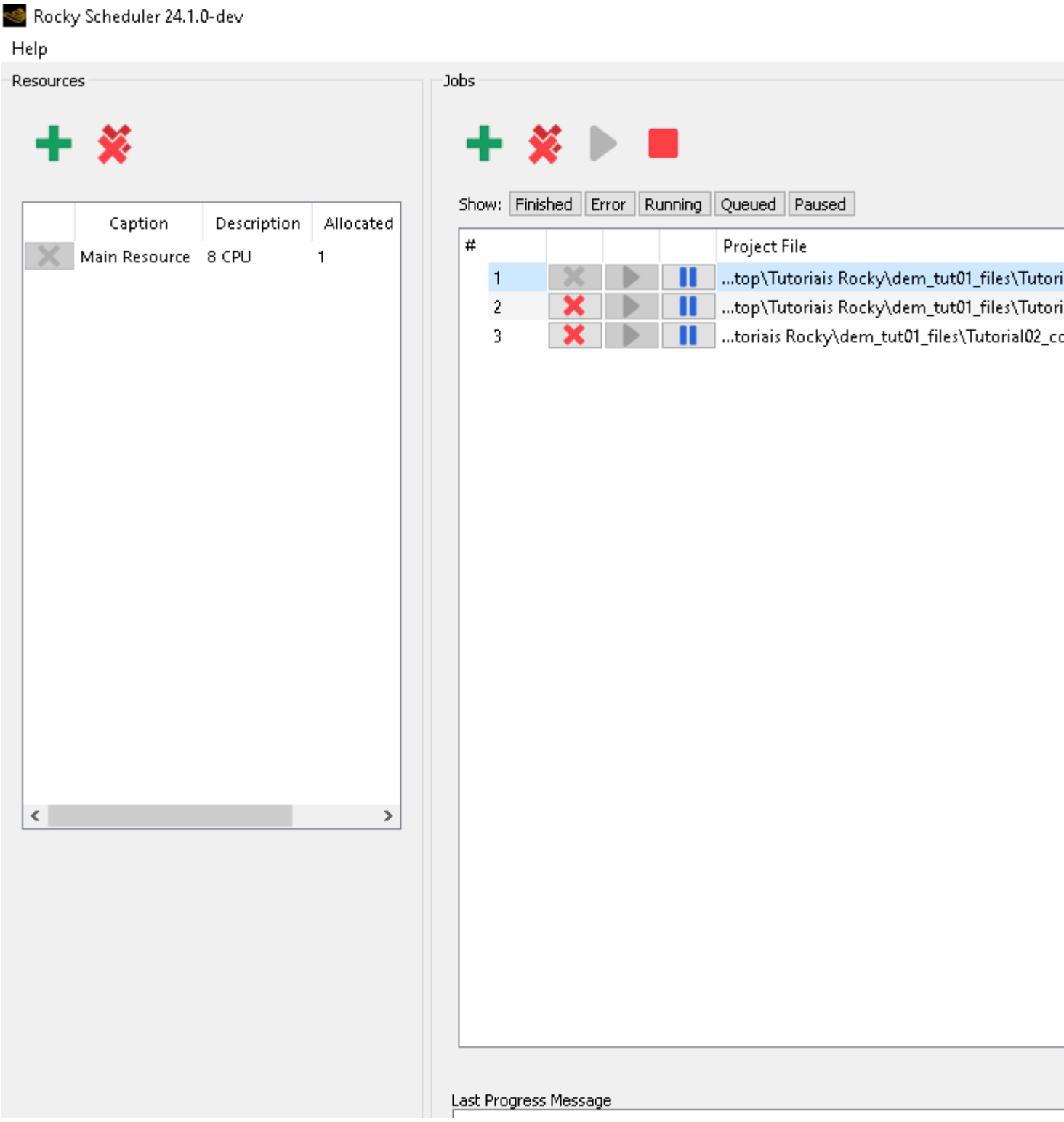
Most users have a single-instance Rocky license and can therefore process only one job at a time (Figure 1). Only those users with unnumbered Rocky licenses, which allows multiple instances of the Rocky solver to be used simultaneously, can use multiple resources to process many jobs at the same time (Figure 2). Rocky Scheduler currently does not make any validations about the license, so if you assign a set of resources that are incompatible with your license, jobs will still try to run but might result in errors. If you are not sure what kind of Rocky license you have, please Submit a Support Request (p. 1293) or contact your Rocky representative for assistance.

Important When choosing to processes very long or complex cases over a network, it is important to periodically check the status of your simulation to ensure that loss of connectivity or other issues haven't stopped it from processing. (See also I get a "Failed to write the log" message while processing my simulation (p. 1340).)

Use the figures and tables below to understand how to set up and use the Rocky Scheduler.

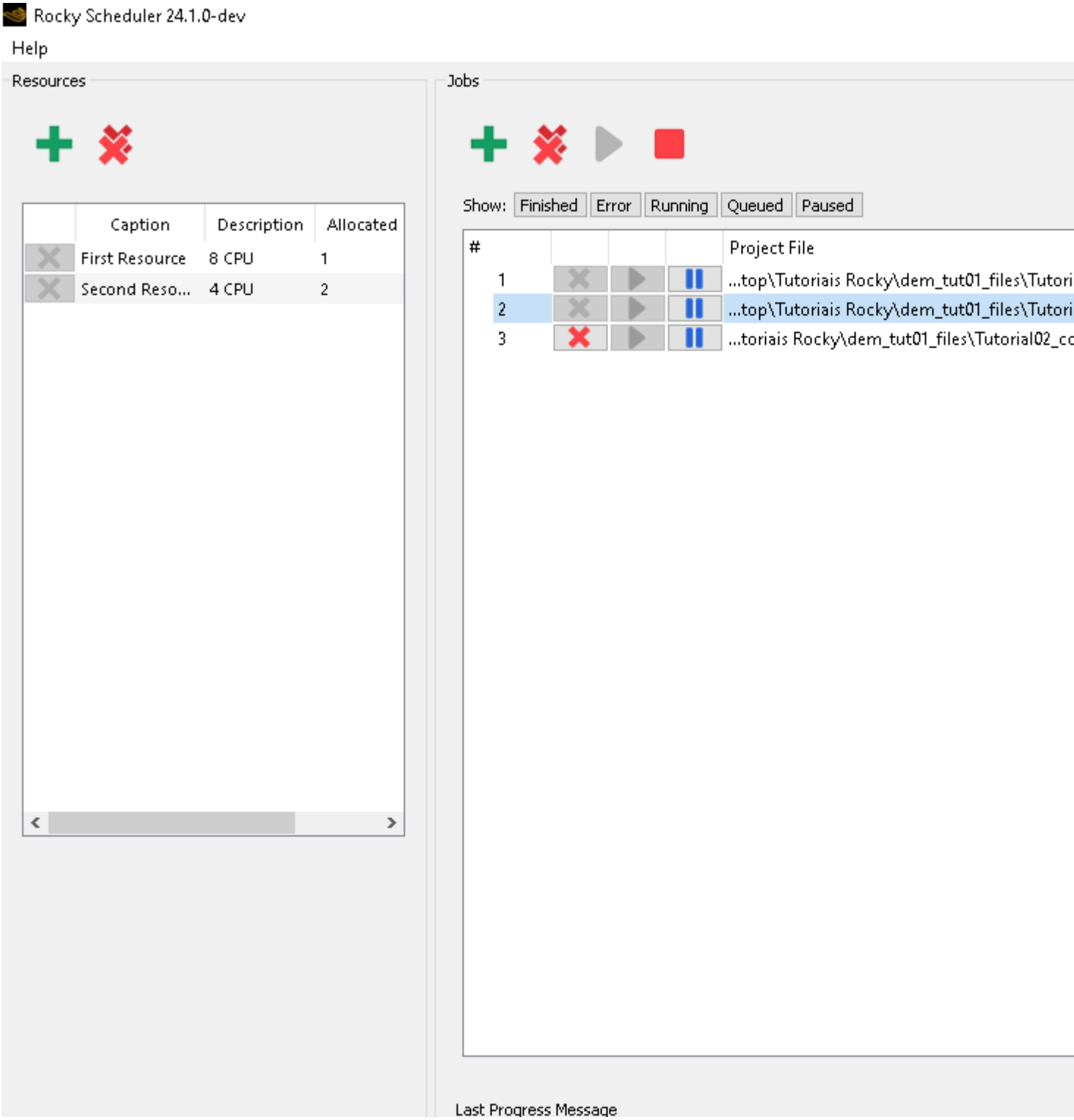
Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

Figure 4.2: Rocky Scheduler with one case running and two in the queue (single-instance license)



Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

Figure 4.3: Rocky Scheduler with 2 cases running simultaneously and one in the queue (unnumbered license)



Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

Figure 4.4: Add Resource dialog for CPU Resource Type

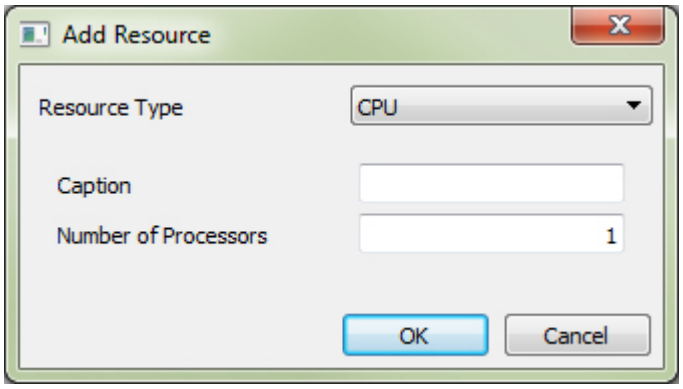


Figure 4.5: Add Resource dialog for a single GPU Resource Type

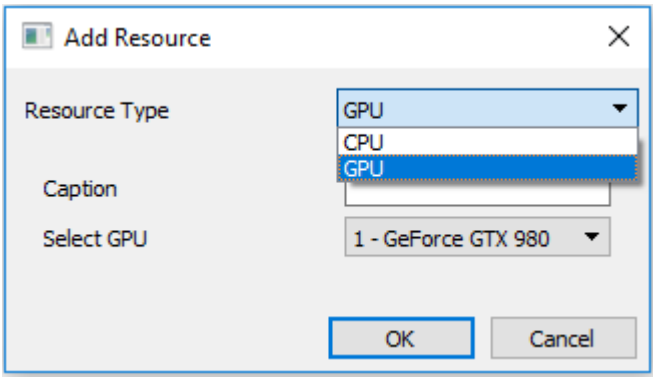
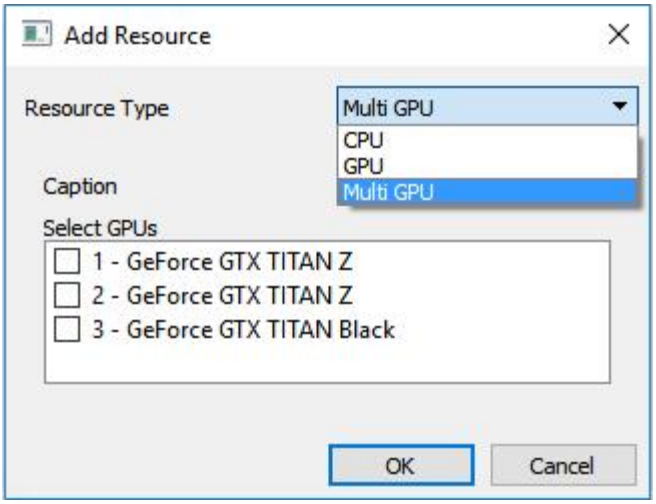


Figure 4.6: Add Resource dialog for multiple GPUs Resource Type



Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

Figure 4.7: Assign Resources dialog

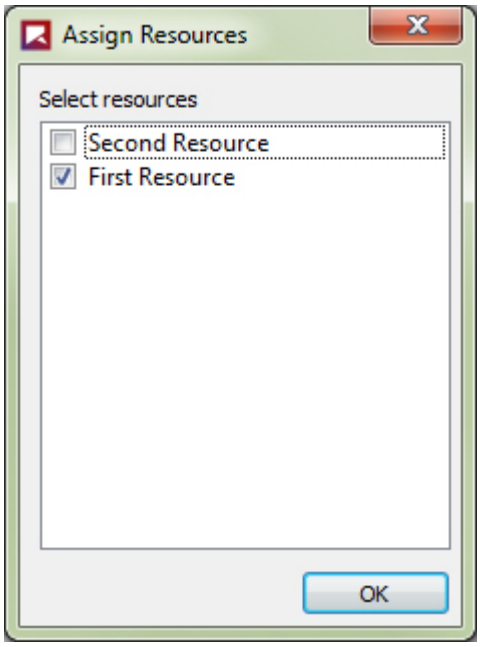


Table 1: Add Resources dialog options

Setting	Description	Range
Resource Type	Enables you to select what computer resource, including CPU and GPU(s), to use for processing.	List automatically generated based upon your computer's resources
Caption	Enables you to provide a unique name for the resource so that you can apply it to the jobs you want later.	Any value
Number of Processors	When CPU is set for **Resource Type** this sets the number of computer processors (threads) the resource will use. In general, the more processors used, the faster the simulation will complete. It is recommended that you set this number equal to the amount of available cores (both physical and virtual) for which you have Rocky licenses. Notes: • Setting more processors than what you have available could slow the simulation. • Windows limits the number of processors to 64 so if you are running Rocky on Windows, it is best to set this value to 64 or less.	Any positive value
Select GPU	When GPU is set for **Resource Type**, this enables you select a single GPU that the resource will use.	List automatically generated based

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

Setting	Description	Range
		upon your computer's resources
Select GPUs	When Multi GPU is set for **Resource Type**, this enables you select one or more GPUs that the resource will use.	List automatically generated based upon your computer's resources

Table 2:Rocky Scheduler Column Descriptions

Column	Description
Resources	
Remove Resource	Provides a button that when clicked, removes the resource from the group of available resources. **Tip:** To clear the entire Resources list, click the **Remove All Resources** button.
Caption	Lists the unique name you provided for the resource.
Description	Lists the number of cores and the resource type (CPU or GPU) that you set.
Allocated by	Lists the job number (**#**) by which the resource is currently being used.
Jobs	
#	The sequential number assigned to the job by the Rocky Scheduler program. This number will determine the order in which the jobs are processed based upon resource availability and license type, and will be referenced in the **Allocated by** column.
Remove Job	Provides a button that when clicked, removes the job from the group of jobs to be processed. **Note:** You can only remove jobs that are not currently being processed. To remove a job that is currently being processed, first click the **Pause Simulation** button, and then click the **Remove Job** button. **Tip:** To clear the entire Jobs list, click the **Remove All Jobs** button.
Project File	Location and name of the .rocky project file set to be processed.

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

Column	Description
Resource	When clicked, enables you to assign one or more resources to use when processing the project. Notes: • When multiple resources are selected, the first available resource will be used to process the simulation. Only one resource can be used per job. • You must select at least one resource for each job. • The resources you can use at any one time are limited by your Rocky license as follows: – Single-instance license users are limited to processing with one resource at a time. – Unnumbered license users can have as many jobs processing at one time as there are resources available. **Tip:** To assign the same Resource to multiple jobs, first multi-select the jobs you want (hold Shift or Ctrl and then click), and then holding the Shift button, double-click one of the selected **Click here to assign resources** bars. **Note:** Even when you choose to process your project with one or more GPUs, Rocky will still use at least 1 CPU core to complete certain pre-processing tasks related to Particle Inputs. After these tasks are complete, Rocky will process the simulation using only the resources you assigned.
Resume	When selected, enables a partially processed simulation to begin processing from the last saved point. When cleared, simulations that have been paused will start processing over from the beginning.
Progress	Shows the processing status and percentage complete of the project file. • Shows "QUEUED" if the file is not currently being processed but will once the resource is available. • Shows "RUNNING" when the file is currently being processed. • Shows "PAUSED" when the file has been partially run but isn't processing currently. • Shows "FINISHED" when processing is complete.

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

Column	Description
	• Shows "ERROR" when processing can't continue due to errors. Notes: – A "No Solver license available" error might mean that you are attempting to run more than one simulation simultaneously with a single-instance license. To resolve this, assign only one resource and run only one job at a time. – A "Failed to write the log" message might mean that something prevented Rocky from being able to save additional simulation files to the location specified, and processing was stopped to avoid loss of simulation data. To resolve this, check your network connection and the storage capacity of your target drive, and then resume processing your simulation. **Tip:** To filter the Jobs list by progress type, click the button you want after **Show**. To turn off a filter, click the same button again.
Message	Shows progress details similar to the Rocky Title bar or error details if the simulation cannot continue.
Last Progress Message	Shows the most recent progress message or error information for the selected job.

What would you like to do?

- Open the Rocky Scheduler Program (p. 927)
- Add a New Resource to the Rocky Scheduler (p. 928)
- Remove a Resource from the Rocky Scheduler (p. 928)
- Process Multiple Simulation Projects In Succession (p. 929)

See Also:

- Processing a Simulation (p. 907)

4.4.2. Open the Rocky Scheduler Program

Note:

Only one instance of the Rocky Scheduler program can be open at one time.

Instructions for opening the Rocky Scheduler program on both Windows- and Linux-based platforms are provided below.

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

To open the Rocky Scheduler Program [Windows]:

From the Programs list on the Windows **Start** menu, click **Ansys 2023 R1**, and then click **Rocky Scheduler 2023 R1**.

To open the Rocky Schedule Program [Linux]:

Do one of the following:

- From a Linux command line, navigate to the same folder from which you run Rocky, and then run `./RockyScheduler`
- From a Linux file explorer, navigate to the same folder from which you run Rocky, and then double-click the `RockyScheduler` executable file.

See Also:

- About Processing Multiple Simulations In Succession (p. 920)
- Add a New Resource to the Rocky Scheduler (p. 928)
- Remove a Resource from the Rocky Scheduler (p. 928)
- Process Multiple Simulation Projects In Succession (p. 929)

4.4.3. Add a New Resource to the Rocky Scheduler

1. Ensure the Rocky Scheduler is open. (See also Open the Rocky Scheduler Program (p. 927).)
2. From the **Resources** section, click **Add Resource**.
3. From the **Add Resource** dialog, choose the **Resource Type**, enter the **Caption**, and then do one of the following:

- For CPU **Resource Type**, enter the **Number of Processors** you want to use.
- For GPU **Resource Type**, select a single GPU from the **Select GPU** list.
- For Multi GPU **Resource Type**, select one or more GPUs from the **Select GPUs** list.

4. Click **OK**. The new resource shows in the Resources list.

See Also:

- About Processing Multiple Simulations In Succession (p. 920)
- Remove a Resource from the Rocky Scheduler (p. 928)
- Process Multiple Simulation Projects In Succession (p. 929)

4.4.4. Remove a Resource from the Rocky Scheduler

1. Ensure the Rocky Scheduler is open. (See also Open the Rocky Scheduler Program (p. 927).)

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

2. Ensure that any jobs assigned to the resource you want to remove are not currently processing by doing one of the following:
   - To stop an individual job from processing, click the **Pause Simulation** button for that job.
   - To stop all jobs from processing, click the **Stop Scheduler** button.
3. From the **Resources** section, click the **Remove Resource** button for the resource you want to remove.

Tip:

To clear the entire Resources list, click the **Remove All Resources** button.

See Also:

- About Processing Multiple Simulations In Succession (p. 920)
- Add a New Resource to the Rocky Scheduler (p. 928)
- Process Multiple Simulation Projects In Succession (p. 929)

4.4.5. Process Multiple Simulation Projects In Succession

Note:

Most users have single-instance Rocky licenses and will therefore process jobs with the Rocky Scheduler sequentially, or one at a time. Only those users with unnumbered Rocky licenses will be able to use more than one resource to process multiple jobs in the Rocky Scheduler simultaneously.

---

1. Ensure the Rocky Scheduler is open. (See also Open the Rocky Scheduler Program (p. 927).)
2. Ensure your **Resources** list is set the way you want. (See also Add a New Resource to the Rocky Scheduler (p. 928).)
3. Under the **Jobs** section, click the **Add Job** button.
4. From the **Choose Rocky project file** dialog, locate and select one or more .rocky project files you want to process, and then click **Open**. Each file you selected appears in the list as a new job row.

Tip:

Add jobs in the order you want the Rocky Scheduler to process them.

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

1. Under the **Resource** column for the job you just added, click the **Click here to assign resources** bar.

Tip:

To assign the same Resource to multiple jobs, first multi-select the jobs you want (hold Shift or Ctrl and then click), and then holding the Shift button, double-click one of the selected **Click here to assign resources** bars.

---

1. From the **Assign Resources** dialog, select one or more resources you want to use to process the selected job(s), and then click **OK**.

Important:

Assign only the amount of resources that are compatible with your Rocky license. Otherwise, jobs will still try to process but may result in errors. (See also About Processing Multiple Simulations In Succession (p. 920).) If you are not sure what kind of Rocky license you have, please Submit a Support Request (p. 1293) or contact your Rocky representative for assistance. Note: When multiple resources are selected, the first resource in the list that becomes available will do the processing.

---

1. Repeat steps 3-6 for each of the projects you want to process.

2. Click the **Start Scheduler** button.

If you have a single-instance Rocky license (most users), then the first job with available resources begins processing. If you have an unnumbered Rocky license, then all jobs with available resources begin processing at the same time.

Tips:

- Press a job's **Pause Simulation** button to prevent that job from processing when it's scheduled to process.
- Press a job's **Start Simulation** button to resume processing a paused simulation when it's scheduled to process.
- Press the **Stop Scheduler** button to stop all jobs from processing.
- After clicking the **Start Scheduler** button, you can minimize the Rocky Scheduler program to work on other tasks.

See Also:

- About Processing Multiple Simulations In Succession (p. 920)
- Remove a Resource from the Rocky Scheduler (p. 928)

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

Chapter 5: Analyzing a Simulation

In Rocky, you can choose to analyze your simulation results by creating 3D Views, viewing individual statistics, and by graphing sets of data in plots and histograms. While you can set up many of these items before you begin processing, it is during processing and after your simulation is complete that these items become most useful. For example, after the processing is complete, you can export data sets and 3D View images —as well as create and save animations of your simulations— for review and sharing outside Rocky

The basic building blocks upon which Rocky enables you to display and analyze data are the various properties presented on the Properties and Curves tabs. Rocky provides these properties as a starting point for creating your desired 3D Views, plots, and histograms.

Rocky also helps you filter the view you want to see and the data you want to analyze by enabling you to define User Processes. These analysis shapes, planes, and settings are the methods by which you divide, reduce, or filter the simulation area to better control the results you see and data you graph.

You change the time step you want to visualize in 3D Views and animations, and what values you want to see for individual Properties and Curves properties on the Time toolbar.

And because more complex analyses may take more time to calculate and display, Rocky provides a Progress panel that you can use to monitor your various requests and determine how long they might take to complete.

What would you like to do?

- Use Properties and Curves (p. 931)
- Create and Modify a 3D View (p. 1040)
- View and Save Simulation Statistics (p. 1097)
- Create and Save an Animation (p. 1166)
- Filter Views and Data with User Processes (p. 1174)
- Change the Current Time Step by Using the Time Toolbar (p. 1278)
- Monitor Analyses Statuses by Using the Progress Panel (p. 1281)

5.1. Use Properties and Curves

The properties presented on the Properties and Curves tabs represent the many different statistics that Rocky records at each time step for each individual element in a structure (e.g., particle, air flow node, geometry triangle) as your simulation processes. The large amount of data that Rocky collects is then distilled into default properties that are accessible from either the Properties or Curves tab for the entity of interest.

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

Once a property is selected, you can quickly display individual statistics, visualize the data set in a 3D View, graph them in a plot or histogram, or export them in tabular format. You may also customize the default properties Rocky provides to create your own unique data sets, including limiting properties by a specific time range by using Time Statistics.

Note:

The process for customizing default properties for Eulerian Statistics User Processes is slightly different from other entities. (See also About Customizing Eulerian Statistics Properties (p. 1017).)

If you need to distill a data set from a property into one final value, or analyze the components that make up your data set, you can do so from the Output tab of the Expressions/Variables panel (see also About Defining Output Variables (p. 250)).

To help you choose the correct property, Rocky separates them into the following two categories:

- **Properties** contain data sets that include recorded values for each individual element in a structure at each individual time step during the simulation. This has the potential of producing many unique values per time step.
- **Curves** contain data sets that apply to the entire entity as a whole, producing only one value for each property.

Note:

For any particles that you have defined as Multiple Element (meshed (p. 1392)), what you have set for **Meshed Particles Upscaling** affects whether and how some post-processing properties and curves are displayed (see also About Meshed Particles Upscaling (p. 1033)).

What would you like to do?

- Learn more About Properties (p. 933)
- Learn more About Curves (p. 989)
- Learn more About Customizing Properties or Curves (p. 1012)
- Learn more About Customizing Eulerian Statistics Properties (p. 1017)
- Learn more About Adding and Editing Time Statistics Properties (p. 1023)
- Learn more About Meshed Particles Upscaling (p. 1033)
- Add a New Custom Property on the Properties or Curves Tab (p. 1037)
- Change or Remove an Existing Custom Property on the Properties or Curves Tab (p. 1036)
- Add a Particle Property to a Eulerian Statistics User Process (p. 1037)
- Add a Time Statistics Property (p. 1039)
- Edit or Remove a Time Statistics Property (p. 1039)

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

See Also:

- Analyzing a Simulation (p. 931)

5.1.1. About Properties

Properties are data sets that include recorded values for each individual element in a structure (e.g., particle, geometry triangle, or air flow node) at each individual *output time* (p. 1393) during the simulation.

Limiting and Displaying Properties

Because they have the potential to produce many unique values for each *output time* (p. 1393), Properties must first be limited by some factor (e.g., showing the data for only one output at a time, showing only the minimum or average value for all output times, and so on) before displaying it in the Rocky UI.

The ways in which you can limit and display your Properties are described in Table 1.

Table 1: Methods for Limiting What a Property will Display

Limit By	Method	See Also
Operations		
By operations only	Create a Time Plot from the property and then choose to limit the values by one of the operations provided, which include Min, Max, Sum, Sum Squared, Average, Variance, and Std. Deviation.	About Time Plots (p. 1102)
By both *output time* (p. 1393) and operations	From the property, choose to Compute Statistics for a single output time and then use the UI to display the individual values you want. The values you can display include the same operations used to limit a Time Plot (Min, Max, and so on) but also include Name, Unit, Type, and Location.	View an Individual Statistic (p. 1098)
Output Time Only		
By creating a 3D View window	Create a 3D View from the function and then use the Time toolbar to walk through the data collected for each individual output time (see also About the Time Toolbar (p. 1279)).	Create and Modify a 3D View (p. 1040)
By creating a Particles Details window	For Intra-Particle Collision Statistics properties only (see also About Intra-Particle Collision Statistics (p. 324)), create a Particles Details window from the function and then use the Time toolbar to walk through the data collected for each individual output time (see also About the Time Toolbar (p. 1279)).	Create and Modify a Particles Details Window (p. 679)
By creating a Histogram	Create a Histogram from the property and then use the Time toolbar to walk through the data collected for each individual output time (see also About the Time Toolbar (p. 1279)).	About Histograms (p. 1112)

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

Calculating Properties

Rocky calculates Properties for individual geometry components, including conveyors and imported geometries; Particles, Contacts, 1-Way LBM, 1-Way Fluent, 2-Way Fluent, Eulerian Statistics, various kinds of Collision Statistics, and individual User Processes based upon these entities (p. 1395).

Note:

When displaying Properties for a User Process, the properties displayed will be those belonging to the linked entity upon which the User Process was based (see also See Input/Output Relationships in the Data Panel When Component Names are Bold (p. 114)).

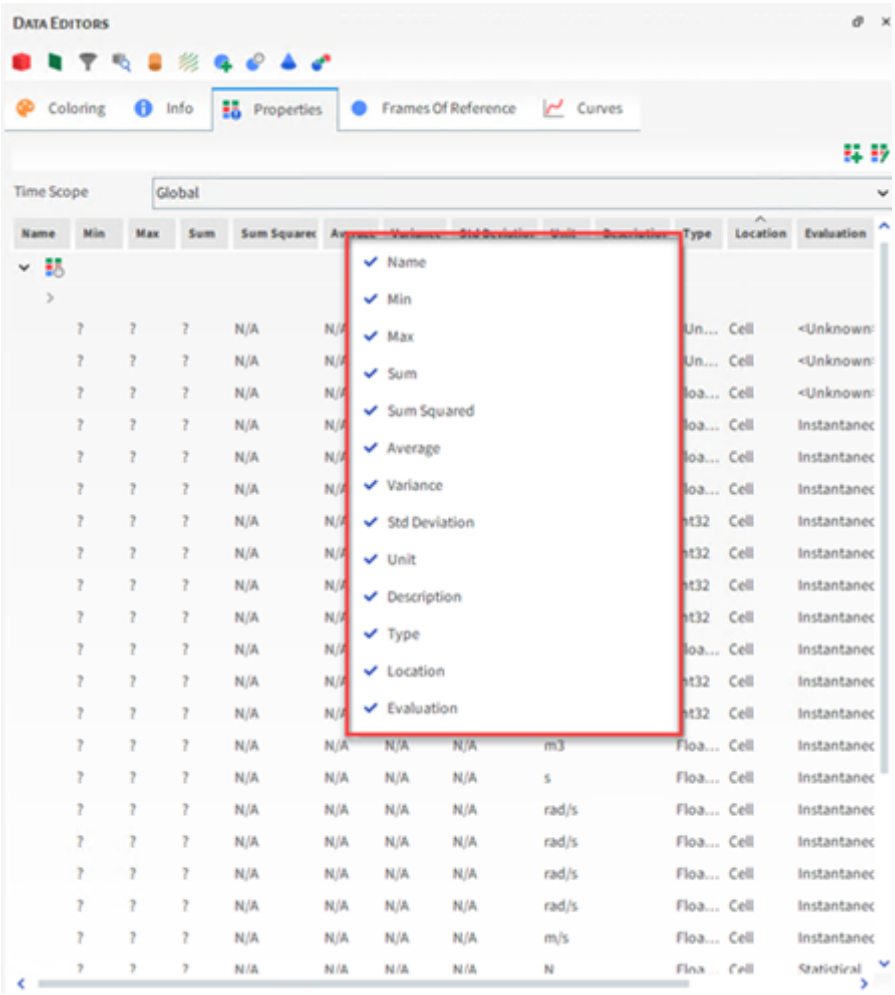
If these default properties do not meet your needs, you may also create custom Properties (see also About Customizing Properties or Curves (p. 1012)), including Time Statistics Properties, which limits properties by a time range (see also About Adding and Editing Time Statistics Properties (p. 1023)).s

Viewing Properties

A list of available properties appears for each applicable component after processing via the component's **Properties** tab. Next to each property name row are various columns representing specific categories of operations and other data (Figure 5.1: Example Properties tab showing all available data columns (p. 935)).

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

Figure 5.1: Example Properties tab showing all available data columns



Besides separate columns for all standard operators—which include values for Min, Max, Sum, Sum Squared, Average, Variance, and Std. Deviation—you can also view the property's Units, Type, Location, and Evaluation.

By right-clicking a property row and then selecting **Compute statistics**, you can get values for that property at that particular output (see also About Viewing an Individual Statistic (p. 1098)).

Definitions of Default Rocky Properties

See the images and tables below to learn about the various Properties available by default in Rocky.

TIME SCOPE FOR PROPERTIES

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

Figure 5.2: Time Scope option on the Properties tab for all entities

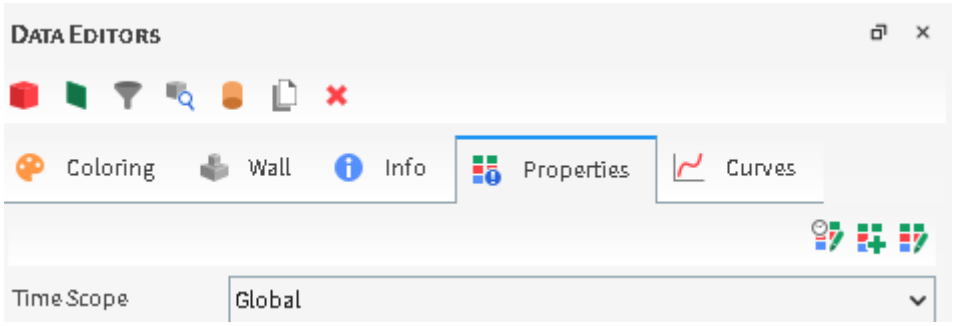


Table 2: Properties available by default for all entities

Property Name	Description
Time Scope	Sets how the resulting statistics are displayed in the various factor columns. Specifically: • **Global** shows statistics that are calculated based upon all available *output time* (p. 1393). • **Instantaneous** shows statistics that are calculated based upon only the selected output time.

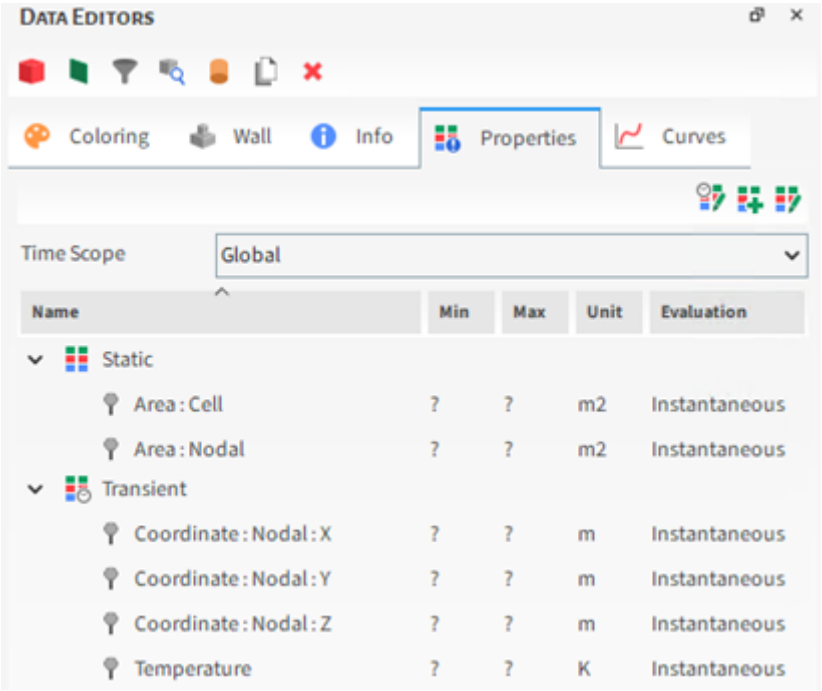
GEOMETRY COMPONENT PROPERTIES

There are two kinds of Properties for individual Geometry components:

- Standard properties, which are calculated by default for every conveyor or imported Geometry (Figure 5.3: Standard Properties options on the Data Editors panel for Geometry components (p. 937) - see also Add and Edit Geometry Components (p. 480)).
- Collision statistics properties, which are collected only when the Boundary Collision Statistics collection feature is enabled (Figure 5.4: Collisions Statistics Properties on the Data Editors panel for Geometry components (p. 938) - see also About Collision Statistics for Boundaries (p. 312)).

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

Figure 5.3: Standard Properties options on the Data Editors panel for Geometry components



Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

Figure 5.4: Collisions Statistics Properties on the Data Editors panel for Geometry components

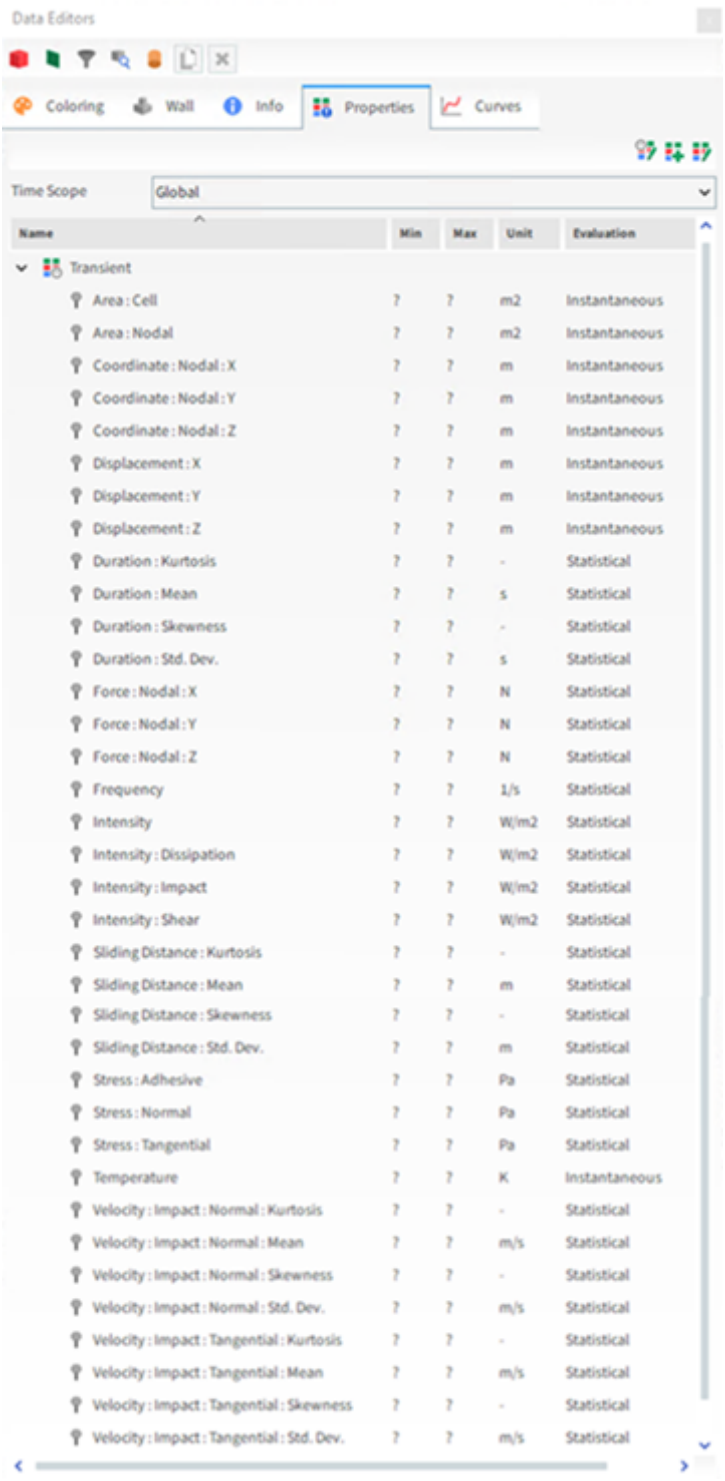


Table 3: Properties available by default for Geometries (Feed Conveyors, Receiving Conveyors, and Imported) (see also I cannot find some Geometry Properties or Curves in this version of Rocky (p. 1369)).

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

Property Name	Description
Standard Properties	
Area : Cell	The surface area of each individual triangle that makes up the selected geometry. For chutes with wear modification parameters, this is useful for estimating how much surface area might be worn away due particle impact (see Enable and View Surface Wear Modification on an Imported Geometry (p. 379)).
Area : Nodal	The area of a region around each individual geometry node. This area is equal to the sum of one third of the area of all triangles that share a given node. This is useful for analyzing the amount of wear on a surface due to shear work.
Coordinate : Nodal : X	The coordinate location along the X axis for each individual geometry node. This is useful for analyzing the amount of shear wear in the X direction.
Coordinate : Nodal : Y	The coordinate location along the Y axis for each individual geometry node. This is useful for analyzing the amount of shear wear in the Y direction.
Coordinate : Nodal : Z	The coordinate location along the Z axis for each individual geometry node. This is useful for analyzing the amount of shear wear in the Z direction.
Displacement : X	For imported geometries with surface wear modification enabled (see also Enable and View Surface Wear Modification on an Imported Geometry (p. 379)), this is the distance that each triangle's vertex was displaced by wear in the X direction.
Displacement : Y	For imported geometries with surface wear modification enabled (see also Enable and View Surface Wear Modification on an Imported Geometry (p. 379)), this is the distance that each triangle's vertex was displaced by wear in the Y direction.
Displacement : Z	For imported geometries with surface wear modification enabled (see also Enable and View Surface Wear Modification on an Imported Geometry (p. 379)), this is the distance that each triangle's vertex was displaced by wear in the Z direction.
Temperature	When **Thermal Model** is enabled (see also Enable Thermal Modeling Calculations (p. 400)), this is the thermodynamic temperature of each individual triangle that makes up the selected geometry.
Boundary Collision Statistics Properties	
Duration : Kurtosis	When the **Duration** option is enabled on the **Boundary Collision Statistics** module (see also About Collision Statistics for Boundaries (p. 312)), this provides the Kurtosis value of the duration of the collisions recorded in different regions of the geometry during an interval between two consecutive *output time* (p. 1393). The kurtosis is a measure of how tall and sharp the central peak of a statistical distribution is. See also the "Collision statistics" section of the DEM Technical Manual (from the Rocky **Help** menu, point to **Manuals**, and then click **DEM Technical Manual**).

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

Property Name	Description
Duration : Mean	When the **Duration** option is enabled on the **Boundary Collision Statistics** module (see also About Collision Statistics for Boundaries (p. 312)), this provides the mean value of the duration of the collisions recorded in different regions of the geometry, during an interval between two consecutive output times. See also the "Collision statistics" section of the DEM Technical Manual (from the Rocky **Help** menu, point to **Manuals**, and then click **DEM Technical Manual**).
Duration : Skewness	When the **Duration** option is enabled on the **Boundary Collision Statistics** module (see also About Collision Statistics for Boundaries (p. 312)), this provides the skewness value of the duration of the collisions recorded in different regions of the geometry, during an interval between two consecutive output times. The skewness is a measure of the asymmetry of a statistical distribution about its mean. See also the "Collision statistics" section of the DEM Technical Manual (from the Rocky **Help** menu, point to **Manuals**, and then click **DEM Technical Manual**).
Duration : Std. Dev.	When the **Duration** option is enabled on the **Boundary Collision Statistics** module (see also About Collision Statistics for Boundaries (p. 312)), this provides the standard deviation value of the duration of the collisions recorded in different regions of the geometry, during an interval between two consecutive output times. See also the "Collision statistics" section of the DEM Technical Manual (from the Rocky **Help** menu, point to **Manuals**, and then click **DEM Technical Manual**).
Force : Nodal : X	When the **Forces for FEM Analysis** option is enabled on the **Boundary Collision Statistics** module (see also About Collision Statistics for Boundaries (p. 312)), this provides the average force in the X direction for each individual geometry node. See also the "Collision statistics" section of the DEM Technical Manual (from the Rocky **Help** menu, point to **Manuals**, and then click **DEM Technical Manual**).
Force : Nodal : Y	When the **Forces for FEM Analysis** option is enabled on the **Boundary Collision Statistics** module (see also About Collision Statistics for Boundaries (p. 312)), this provides the average force in the Y direction for each individual geometry node. See also the "Collision statistics" section of the DEM Technical Manual (from the Rocky **Help** menu, point to **Manuals**, and then click **DEM Technical Manual**).
Force : Nodal : Z	When the **Forces for FEM Analysis** option is enabled on the **Boundary Collision Statistics** module (see also About Collision Statistics for Boundaries (p. 312)), this provides the average force in the Z direction for each individual geometry node.

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

Property Name	Description
	See also the "Collision statistics" section of the DEM Technical Manual (from the Rocky **Help** menu, point to **Manuals**, and then click **DEM Technical Manual**).
Frequency	When the **Frequency** option is enabled on the **Boundary Collision Statistics** module (see also About Collision Statistics for Boundaries (p. 312)), this provides the average collision frequency recorded in different regions of the geometry, during an interval between two consecutive output times. See also the "Collision statistics" section of the DEM Technical Manual (from the Rocky **Help** menu, point to **Manuals**, and then click **DEM Technical Manual**).
Intensity	When the **Intensities** option is enabled on the **Boundary Collision Statistics** module (see also About Collision Statistics for Boundaries (p. 312)), this, for each triangle on a boundary, provides the average power needed to move it with the prescribed velocity, divided by the triangle's area. See also the "Collision statistics" section of the DEM Technical Manual (from the Rocky **Help** menu, point to **Manuals**, and then click **DEM Technical Manual**).
Intensity : Dissipation	When the **Intensities** option is enabled on the **Boundary Collision Statistics** module (see also About Collision Statistics for Boundaries (p. 312)), this provides the average power dissipated by the contact forces, divided by the triangle's area, measured by each individual geometry triangle. See also the "Collision statistics" section of the DEM Technical Manual (from the Rocky **Help** menu, point to **Manuals**, and then click **DEM Technical Manual**).
Intensity : Impact	When the **Intensities** option is enabled on the **Boundary Collision Statistics** module (see also About Collision Statistics for Boundaries (p. 312)), this provides the average impact power measured by each individual triangle and divided by the triangle's area. This is useful for determining impact wear on geometry surfaces. See also the "Collision statistics" section of the DEM Technical Manual (from the Rocky **Help** menu, point to **Manuals**, and then click **DEM Technical Manual**).
Intensity : Shear	When the **Intensities** option is enabled on the **Boundary Collision Statistics** module (see also About Collision Statistics for Boundaries (p. 312)), this provides the average shear power measured by each individual triangle. This is useful for determining shear wear on geometry surfaces. See also the "Collision statistics" section of the DEM Technical Manual (from the Rocky **Help** menu, point to **Manuals**, and then click **DEM Technical Manual**)
Sliding Distance : Kurtosis	When the **Sliding Distance** option is enabled on the **Boundary Collision Statistics** module (see also About Collision Statistics for Boundaries (p. 312)), this provides the Kurtosis value of the sliding distance, which is the distance that a particle moves during a collision, parallel to the boundary triangle plane

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

Property Name	Description
	where the collision occurs. This is useful for determining boundary wear caused by shear. See also the "Collision statistics" section of the DEM Technical Manual (from the Rocky **Help** menu, point to **Manuals**, and then click **DEM Technical Manual**).
Sliding Distance : Mean	When the **Sliding Distance** option is enabled on the **Boundary Collision Statistics** module (see also About Collision Statistics for Boundaries (p. 312)), this provides the mean value of the sliding distance, which is the distance that a particle moves during a collision, parallel to the boundary triangle plane where the collision occurs. This is useful for determining boundary wear caused by shear. See also the "Collision statistics" section of the DEM Technical Manual (from the Rocky **Help** menu, point to **Manuals**, and then click **DEM Technical Manual**).
Sliding Distance : Skewness	When the **Sliding Distance** option is enabled on the **Boundary Collision Statistics** module (see also About Collision Statistics for Boundaries (p. 312)), this provides the Skewness value of the sliding distance, which is the distance that a particle moves during a collision, parallel to the boundary triangle plane where the collision occurs. This is useful for determining boundary wear caused by shear. See also the "Collision statistics" section of the DEM Technical Manual (from the Rocky **Help** menu, point to **Manuals**, and then click **DEM Technical Manual**).
Sliding Distance : Std. Dev.	When the **Sliding Distance** option is enabled on the **Boundary Collision Statistics** module (see also About Collision Statistics for Boundaries (p. 312)), this provides the standard deviation value of the sliding distance, which is the distance that a particle moves during a collision, parallel to the boundary triangle plane where the collision occurs. This is useful for determining boundary wear caused by shear. See also the "Collision statistics" section of the DEM Technical Manual (from the Rocky **Help** menu, point to **Manuals**, and then click **DEM Technical Manual**).
Stress : Adhesive	When the **Stresses** option is enabled on the **Boundary Collision Statistics** module (see also About Collision Statistics for Boundaries (p. 312)), an **Adhesive Force** model (other than **None**) is enabled (see also About Physics Parameters (p. 417)), and an Adhesive **Force Fraction** for a boundary-boundary and/or boundary-particle Materials Interaction pair has a value set higher than zero (see also About Modifying Materials Interactions and Adhesion Values (p. 597)), this provides the average adhesive stress in the normal direction, as measured by each individual geometry triangle. See also the "Collision statistics" section of the DEM Technical Manual (from the Rocky **Help** menu, point to **Manuals**, and then click **DEM Technical Manual**).

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

Property Name	Description
Stress : Normal	When the **Stresses** option is enabled on the **Boundary Collision Statistics** module (see also About Collision Statistics for Boundaries (p. 312)), this provides the average normal stress measured by each individual geometry triangle. See also the "Collision statistics" section of the DEM Technical Manual (from the Rocky **Help** menu, point to **Manuals**, and then click **DEM Technical Manual**).
Stress : Tangential	When the **Stresses** option is enabled on the **Boundary Collision Statistics** module (see also About Collision Statistics for Boundaries (p. 312)), this provides the average tangential stress measured by each individual geometry triangle. See also the "Collision statistics" section of the DEM Technical Manual (from the Rocky **Help** menu, point to **Manuals**, and then click **DEM Technical Manual**).
Velocity : Impact : Normal : Kurtosis	When the **Normal Impact Velocity** option is enabled on the **Boundary Collision Statistics** module (see also About Collision Statistics for Boundaries (p. 312)), this provides the kurtosis value of the impact relative velocity in the normal direction resulting from the collisions recorded in different regions of the geometry, during an interval between two consecutive output times. The kurtosis is a measure of how tall and sharp the central peak of a statistical distribution is. See also the "Collision statistics" section of the DEM Technical Manual (from the Rocky **Help** menu, point to **Manuals**, and then click **DEM Technical Manual**).
Velocity : Impact : Normal : Mean	When the **Normal Impact Velocity** option is enabled on the **Boundary Collision Statistics** module (see also About Collision Statistics for Boundaries (p. 312)), this provides the mean value of the impact relative velocity in the normal direction resulting from the collisions recorded in different regions of the geometry, during an interval between two consecutive output times. See also the "Collision statistics" section of the DEM Technical Manual (from the Rocky **Help** menu, point to **Manuals**, and then click **DEM Technical Manual**).
Velocity : Impact : Normal : Skewness	When the **Normal Impact Velocity** option is enabled on the **Boundary Collision Statistics** module (see also About Collision Statistics for Boundaries (p. 312)), this provides the skewness value of the impact relative velocity in the normal direction resulting from the collisions recorded in different regions of the geometry, during an interval between two consecutive output times. The skewness is a measure of the asymmetry of a statistical distribution about its mean. See also the "Collision statistics" section of the DEM Technical Manual (from the Rocky **Help** menu, point to **Manuals**, and then click **DEM Technical Manual**).
Velocity : Impact :	When the **Normal Impact Velocity** option is enabled on the **Boundary Collision Statistics** module (see also About Collision Statistics for

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

Property Name	Description
Normal : Std. Dev.	Boundaries (p. 312)), this provides the standard deviation value of the impact relative velocity in the normal direction resulting from the collisions recorded in different regions of the geometry, during an interval between two consecutive output times. See also the "Collision statistics" section of the DEM Technical Manual (from the Rocky **Help** menu, point to **Manuals**, and then click **DEM Technical Manual**).
Velocity : Impact : Tangential : Kurtosis	When the **Tangential Impact Velocity** option is enabled on the **Boundary Collision Statistics** module (see also About Collision Statistics for Boundaries (p. 312)), this provides the Kurtosis value of the impact relative velocity in the tangential direction resulting from the collisions recorded in different regions of the geometry, during an interval between two consecutive output times. The kurtosis is a measure of how tall and sharp the central peak of a statistical distribution is. See also the "Collision statistics" section of the DEM Technical Manual (from the Rocky **Help** menu, point to **Manuals**, and then click **DEM Technical Manual**).
Velocity : Impact : Tangential : Mean	When the **Tangential Impact Velocity** option is enabled on the **Boundary Collision Statistics** module (see also About Collision Statistics for Boundaries (p. 312)), this provides the mean value of the impact relative velocity in the tangential direction resulting from the collisions recorded in different regions of the geometry, during an interval between two consecutive output times. See also the "Collision statistics" section of the DEM Technical Manual (from the Rocky **Help** menu, point to **Manuals**, and then click **DEM Technical Manual**).
Velocity : Impact : Tangential : Skewness	When the **Tangential Impact Velocity** option is enabled on the **Boundary Collision Statistics** module (see also About Collision Statistics for Boundaries (p. 312)), this provides the skewness value of the impact relative velocity in the tangential direction resulting from the collisions recorded in different regions of the geometry, during an interval between two consecutive output times. The skewness is a measure of the asymmetry of a statistical distribution about its mean. See also the "Collision statistics" section of the DEM Technical Manual (from the Rocky **Help** menu, point to **Manuals**, and then click **DEM Technical Manual**).
Velocity : Impact : Tangential : Std. Dev.	When the **Tangential Impact Velocity** option is enabled on the **Boundary Collision Statistics** module (see also About Collision Statistics for Boundaries (p. 312)), this provides the standard deviation value of the impact relative velocity in the tangential direction resulting from the collisions recorded in different regions of the geometry, during an interval between two consecutive output times.

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

Property Name	Description
	See also the "Collision statistics" section of the DEM Technical Manual (from the Rocky **Help** menu, point to **Manuals**, and then click **DEM Technical Manual**).

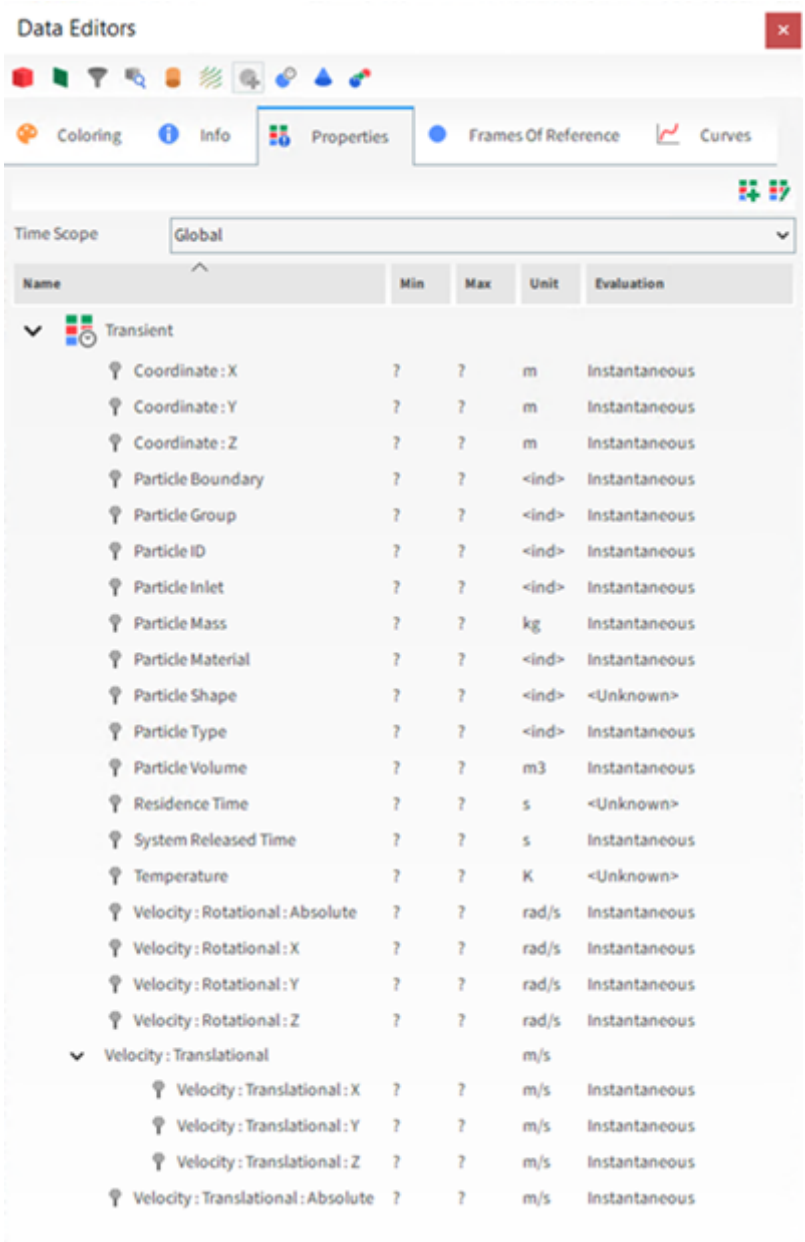
PARTICLES PROPERTIES (SIMULATION-WIDE)

There are several kinds of Properties that can affect all Particles in the simulation. These include:

- Standard properties, which are calculated by default and are applicable to all particle sets in the simulation (Figure 5.5: Particles, Standard Property options in the Data Editors panel (p. 946). See also About Adding and Editing Particle Sets (p. 602)).
- Inter-particle Collision statistics properties, which are collected for all Particles only when the Inter-particle Collision Statistics collection feature is enabled (Figure 5.6: Particles, Property options in the Data Editors panel when Inter-particle Collision Statistics are collected (p. 947). See also About Inter-particle Collision Statistics (p. 331)). Note: Some Inter-particle Collision Statistics properties will not be provided for *meshed* (p. 1392) particles with **Meshed Particles Upscaling** enabled (see also About Meshed Particles Upscaling (p. 1033)).
- Particle Instantaneous Energies properties, which are collected for all Particles only when the Particle Instantaneous Energies collection feature is enabled (Figure 5.7: Particles, Property options in the Data Editors panel when Particle Instantaneous Energies are collected (p. 948). See also About Particle Instantaneous Energies (p. 341)).
- Coarse Grain Modeling (CGM) properties, which appear for all Particles only when the CGM model is enabled (see also About Physics Parameters (p. 417)).
- CFD Coupling Particle Statistics properties, which are collected for all Particles only when the CFD Coupling Particle Statistics collection feature is enabled (Figure 5.8: Particles, Property options in the Data Editors panel when CFD Coupling Particle Statistics are collected (p. 949). See also About Fluid-Related Statistics for Particles (p. 335)).

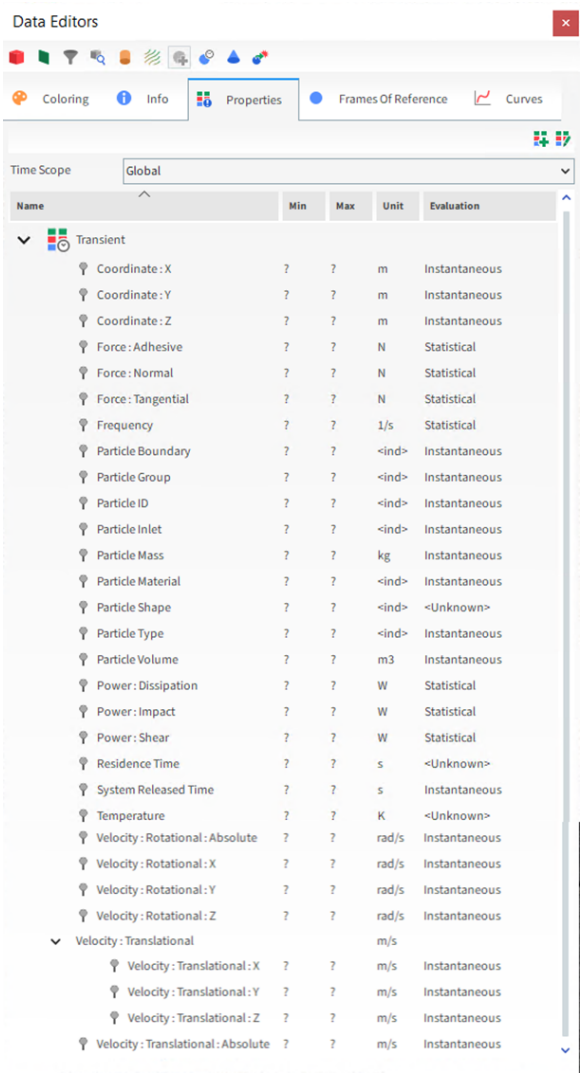
Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

Figure 5.5: Particles, Standard Property options in the Data Editors panel



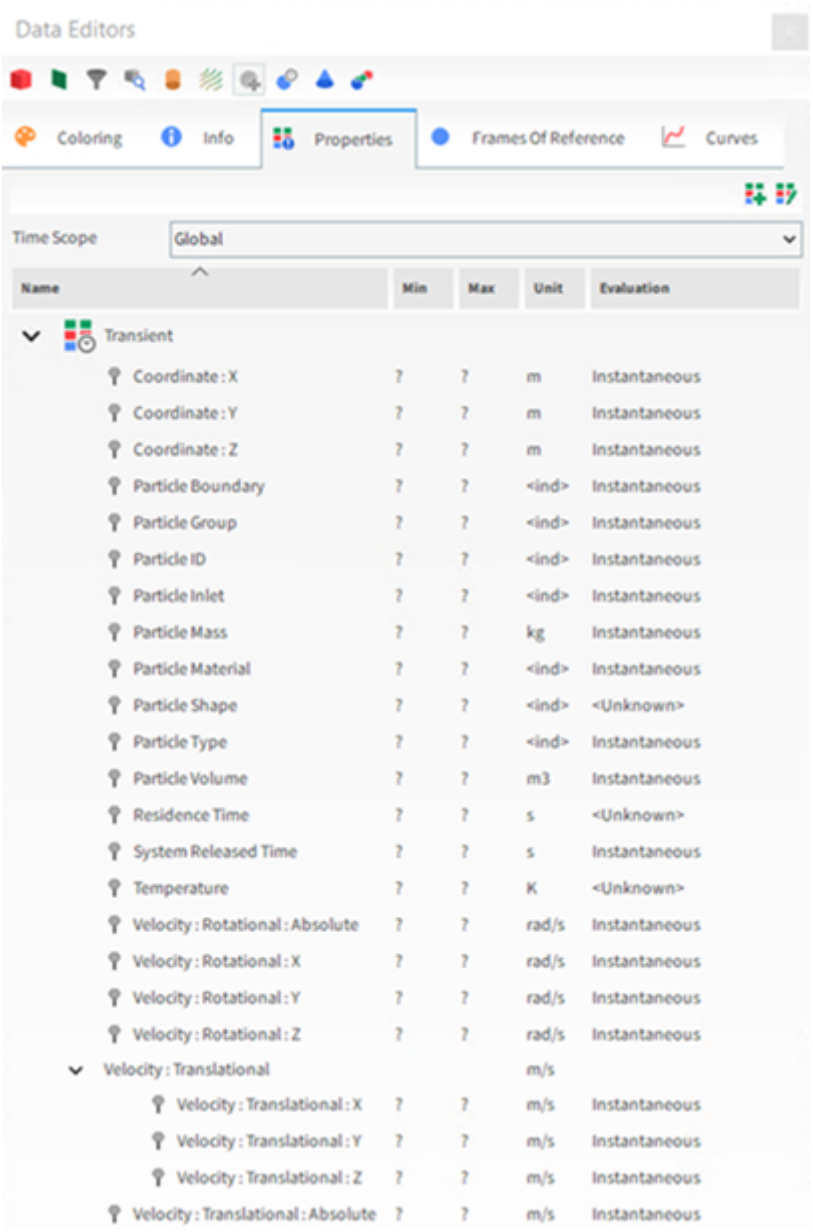
Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

Figure 5.6: Particles, Property options in the Data Editors panel when Inter-particle Collision Statistics are collected



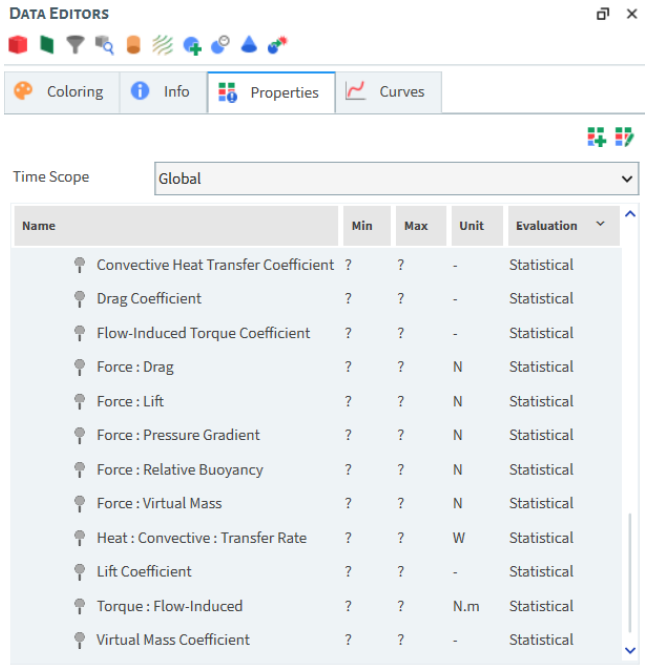
Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

Figure 5.7: Particles, Property options in the Data Editors panel when Particle Instantaneous Energies are collected



Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

Figure 5.8: Particles, Property options in the Data Editors panel when CFD Coupling Particle Statistics are collected



Note: Due to the different ways that Rocky calculates individual particles—for example, based upon whether the particle is whole or broken into *fragment* (p. 1393) (see also Enable Particle Breakage Calculations (p. 397)), remains single or has been *meshed* (p. 1392) into multiple *element* (p. 1390) (see also About Adding and Editing Particle Sets (p. 602)), has particle upscaling effects enabled or disabled (see also About Meshed Particles Upscaling (p. 1033)), and so on—some Properties will not be available for particles that are meshed, or for particles that are calculated with the **Meshed Particles Upscaling** checkbox enabled.

Table 4: Properties available for the main Particles entity

Property Name	**Description**
Standard Properties	
Velocity : Rotational : Absolute	The magnitude of the rotational velocity of each individual whole particle or *fragment* (p. 1393). This is useful for determining the angular momentum of particles, which affects particle rolling. It is defined as the norm of the particle's absolute rotational velocity vector.
Velocity : Translational : Absolute	The magnitude of the translational velocity of each individual whole particle or *fragment* (p. 1393). This is useful for locating stalled particles in chutes, for example. It can be defined as the norm of the particle's absolute translational velocity vector.
Flip Count	For non-flexible, *shaped particle* (p. 1395) only, this is the number of times a whole particle makes a complete, 360 degree turn inside a region. It is useful

Property Name	Description
	for measuring coating efficiency for pills and snack foods (see also About Particles Calculations (p. 1242)). **Note:** This Property is not compatible with Breakage calculations (see also Enable Particle Breakage Calculations (p. 397)) nor with particles composed of multiple *element* (p. 1390) (flexible).
Laguerre-Voronoi Size	When *Instantaneous Breakage* parameters are enabled (see also Enable Particle Breakage Calculations (p. 397)), this is the diameter of a sphere used to generate the *fragment* (p. 1393) using the Laguerre-Voronoi tessellation. It can be defined as twice the minimum distance between the particle's center of mass and the particle's sides: $L_{LV}=2\min[(x_s-x_c)\cdot \mathrm{n}_s$ where: x_s is the position vector of a vertex on a side, x_c is the position vector of the center of mass, and n_s is the unit vector normal to a side. **Note:** This property is not defined for fragments that result from *Discrete Breakage* events.
Moment of Inertia	The moment of inertia of each individual whole particle or fragment. Notes: • This is a tensor property and as such, cannot be made use of in the Rocky UI. However, it is possible for an advanced user to export and then make use of the components (see also I cannot display or plot a particular property (p. 1375)). • The units for this property indicate that it is different from the **Principal Moment of Inertia** value that you might have defined manually for a non-homogeneous particle, which is actually moment of inertia divided by density (see also About Adding and Editing Particle Sets (p. 602)). • Does not apply to *meshed* (p. 1392) particles with particles upscaling enabled (see also About Meshed Particles Upscaling (p. 1033)).
Orientation : Angle	This angle together with the **Orientation Vector** defines the instantaneous orientation of each individual whole particle or fragment.

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

Property Name	Description
	See also the "Particle orientation" section of the DEM Technical Manual (from the Rocky **Help** menu, point to **Manuals**, and then click **DEM Technical Manual**). These properties illustrate how much each particle has rotated and can be useful in pharmaceutical applications where the efficiency of coating is affected by the pill's orientation in relation to the equipment. **Note:** Does not apply to *meshed* (p. 1392) particles with particles upscaling enabled. (See also About Meshed Particles Upscaling (p. 1033).)
Orientation : Vector : X	The x-axis component of a unit vector that defines the axis of rotation, which, together with the **Orientation Angle**, specifies the instantaneous orientation of each individual whole particle or fragment. See also the "Particle orientation" section of the DEM Technical Manual (from the Rocky **Help** menu, point to **Manuals**, and then click **DEM Technical Manual**). **Note:** Does not apply to *meshed* (p. 1392) particles with particles upscaling enabled (see also About Meshed Particles Upscaling (p. 1033)).
Orientation : Vector : Y	The y-axis component of a unit vector that defines the axis of rotation, which, together with the **Orientation Angle**, specifies the instantaneous orientation of each individual whole particle or fragment. See also the "Particle orientation" section of the DEM Technical Manual (from the Rocky **Help** menu, point to **Manuals**, and then click **DEM Technical Manual**). **Note:** Does not apply to *meshed* (p. 1392) particles with particles upscaling enabled (see also About Meshed Particles Upscaling (p. 1033)).
Orientation : Vector : Z	The z-axis component of a unit vector that defines the axis of rotation, which, together with the **Orientation Angle**, specifies the instantaneous orientation of each individual whole particle or fragment.

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

Property Name	Description
	See also the "Particle orientation" section of the DEM Technical Manual (from the Rocky **Help** menu, point to **Manuals**, and then click **DEM Technical Manual**). **Note:** Does not apply to *meshed* (p. 1392) particles with particles upscaling enabled (see also About Meshed Particles Upscaling (p. 1033)).
Parent ID	For simulations with *Instantaneous Breakage* (p. 1392) parameters enabled (see also Enable Particle Breakage Calculations (p. 397)), this index is used to identify the original whole particle or *fragment* (p. 1393) from which the broken fragments were generated. Specifically: • A value of **-1** indicates that the particle is itself a whole particle and has not yet been broken. • A value of **0 or greater** indicates that the fragment was generated from a whole particle or fragment with a **Particle ID** of that value. **Tip:** To identify all the particle fragments that were generated within a simulation, use the Property User Process to select Parent IDs with values equal to and greater than zero (see also About the Property User Process (p. 1202)).
Particle Boundary	The name of the geometry source (i.e., feed conveyor or inlet) through which each individual whole particle originally entered the simulation, and which is then inherited by any fragments that are generated. Particle Boundaries are numbered from zero in (roughly) the order in which they were added.
Particle Equivalent Diameter	The diameter of a sphere with the same volume of the whole particle or fragment. $L_{vol} = \sqrt[3]{\frac{6}{\pi} V}$ **Note:** Does not apply to *meshed* (p. 1392) particles with particles upscaling enabled (see also About Meshed Particles Upscaling (p. 1033)).
Particle Group	The name of the *Particle set* (p. 1393) to which each individual whole particle originally belonged, and which is also inherited by any *fragment* (p. 1393) that

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

Property Name	Description
	are generated. Particle Groups are numbered from zero in (roughly) the order in which they were added. **Tip:** The numbers are needed, in part, when creating a Property User Process (see also About the Property User Process (p. 1202)). Use the Color scale to map the numbers to the Particle Group name (see also About Color Scales (p. 1086)).
Particle ID	The numeric identifier unique to each individual whole particle or fragment.
Particle Inlet	The name of the inlet through which each individual whole particle originally entered the simulation.
Particle Mass	The mass of each individual whole particle or fragment.
Particle Material	The material assigned to each individual whole particle, which is then inherited by any fragments that are generated. By default, this is "Default Particles" (see also About Modifying Material Compositions (p. 591)). Particle Materials are numbered from zero in (roughly) the order in which they were created.
Particle Shape	The **Shape** name category to which each individual whole particle in each *Particle set* (p. 1393) was originally assigned, including its *fragment* (p. 1393), which inherit this value. Particle Shapes are numbered from zero in (roughly) the following order: • Sphere • Polyhedron • Sphero-Cylinder • Sphero-Polygon • Shpero-Polyhedron • Briquette • Faceted Cylinder • Custom Polyhedron • Custom Shell • Straight Fiber

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

Property Name	Description
	• Custom Fiber **Note:** This property replaces the now legacy **Particle Type** property. **Tip:** The numbers are needed, in part, when creating a Property User Process (see also About the Property User Process (p. 1202)). Use the Color scale to map the numbers to the Particle Shape type (see also About Color Scales (p. 1086)).
Particle Size	The sieve size of each individual whole particle or fragment, which is based upon the dimensions of a square hole just big enough for the particle to pass through. This use of virtual "mesh" determines the size of the particles no matter what shape they are. **Note:** Does not apply to *meshed* (p. 1392) particles with particles upscaling enabled (see also About Meshed Particles Upscaling (p. 1033)).
Particle Surface Area	The surface area of each individual whole particle or fragment. **Note:** Applies only to non-*meshed* (p. 1392) particles (see also About Adding and Editing Particle Sets (p. 602)).

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

Property Name	Description
Particle Type (Legacy Use Only)	**Important:** This property does not reflect newer particle types and is retained only for compatibility purposes. For new projects, use the **Particle Shape** Property instead. For projects created prior to Rocky v4.1, this is the shape type assigned to each individual whole particle, which is then inherited by any *fragment* (p. 1393) that are created (see also About Adding and Editing Particle Sets (p. 602)). The particle type number maps to the particle shapes as follows: 1. Sphere 2. Polyhedron, Custom Polyhedron, Shell, and Fiber shapes 3. Sphero-Cylinder 4. Sphero-Polygon 5. Sphero-Polyhedron 6. Briquette 7. Faceted Cylinder
Particle Volume	The volume of each individual whole particle or fragment.
Coordinate : X	The coordinate location along the X-axis for each individual whole particle or fragment.
Coordinate : Y	The coordinate location along the Y-axis for each individual whole particle or fragment.
Coordinate : Z	The coordinate location along the Z-axis for each individual whole particle or fragment.
Residence Time	The amount of time each individual whole particle or fragment has been within the specified region boundaries (see also About Particles Calculations (p. 1242)). It can be defined as the difference between the current time, t, and the time the particle entered the region, t_{in}. $r_{local}=t-t_{in}$

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

Property Name	Description
	Particle fragments initially inherit this value from their parent upon breaking, but are then calculated separately. For example, if a fragment tumbles out of the selection and then re-enters it later, its Residence Time will increase. **Note:** This Property is not compatible with projects processed with earlier versions of Rocky (prior to Rocky v4) that also had breakage modeling enabled (see also Enable Particle Breakage Calculations (p. 397)).
Velocity : Rotational : X	The x-component of the rotational velocity vector of each individual whole particle or fragment.
Velocity : Rotational : Y	The y-component of the rotational velocity vector of each individual whole particle or fragment.
Velocity : Rotational : Z	The z-component of the rotational velocity vector of each individual whole particle or fragment.
Stress Tensor	When the **Collect Contacts Data** checkbox is enabled (see also About Contacts (p. 746)), this provides the stress tensor of a particle p, which is computed as: $\sigma_p = \frac{1}{V_p} \sum_{c=1}^{N_p^c} l_c \mathrm{F}_c$ where: V_p is the volume of the particle p, l_c is a vector that connects the centroid of the particle p to the contact point c, F_c is a contact force vector at contact point c, and N_p^c is the number of contacts of particle p. Notes: • This property provides data for all particles but the calculations are currently valid only for spherical particle shapes. • This is a tensor property and as such, cannot be made use of in the Rocky UI. However, it is possible for an advanced user to export and then make use of the components (see also I cannot display or plot a particular property (p. 1375)). • Applies only to non-*meshed* (p. 1392) particles (see also About Adding and Editing Particle Sets (p. 602)).
System Released Time	The instance in simulation time when each individual whole particle or fragment originally entered the simulation boundaries.

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

Property Name	Description
Temperature	When **Thermal Model** is enabled (see also Enable Thermal Modeling (p. 400)), this is the thermodynamic temperature of each individual whole particle and fragment.
Velocity : Translational : Absolute	The translational velocity vector of each individual whole particle or fragment, the magnitude of which is made up of the x, y, and z components respectively. **Note:** Because this is a vector, display is limited to the 3D View only. When shown in a 3D view, the particles will be colored by the magnitude of the velocity vector.
Velocity : Translational : X	The x-component of the current translational velocity vector of each individual whole particle or fragment.
Velocity : Translational : Y	The y-component of the current translational velocity vector of each individual whole particle or fragment.
Velocity : Translational : Z	The z-component of the current translational velocity vector of each individual whole particle or fragment.
Inter-Particle Collision Statistics Properties	
Duration : Kurtosis	When the **Duration** option is enabled on the **Inter-particle Collision Statistics** module (see also About Inter-particle Collision Statistics (p. 331)), this provides the kurtosis value of the duration of the collisions recorded by each individual whole particle or fragment during an interval between two consecutive *output time* (p. 1393). The kurtosis is a measure of how tall and sharp the central peak of a statistical distribution is. **Note:** Does not apply to *meshed* (p. 1392) particles with particles upscaling enabled (see also About Meshed Particles Upscaling (p. 1033)). See also the "Collision statistics" section of the DEM Technical Manual (from the Rocky **Help** menu, point to **Manuals**, and then click **DEM Technical Manual**).
Duration : Mean	When the **Duration** option is enabled on the **Inter-particle Collision Statistics** module (see also About Inter-particle Collision Statistics (p. 331)), this provides the mean value of the duration of the collisions recorded by each individual

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

Property Name	Description
	whole particle or fragment, during an interval between two consecutive output times. **Note:** Does not apply to *meshed* (p. 1392) particles with particles upscaling enabled (see also About Meshed Particles Upscaling (p. 1033)). See also the "Collision statistics" section of the DEM Technical Manual (from the Rocky **Help** menu, point to **Manuals**, and then click **DEM Technical Manual**).
Duration : Skewness	When the **Duration** option is enabled on the **Inter-particle Collision Statistics** module (see also About Inter-particle Collision Statistics (p. 331)), this provides the skewness value of the duration of the collisions recorded by each individual whole particle or fragment, during an interval between two consecutive output times. The skewness is a measure of the asymmetry of a statistical distribution about its mean. **Note:** Does not apply to *meshed* (p. 1392) particles with particles upscaling enabled (see also About Meshed Particles Upscaling (p. 1033)). See also the "Collision statistics" section of the DEM Technical Manual (from the Rocky **Help** menu, point to **Manuals**, and then click **DEM Technical Manual**).
Duration : Std. Dev.	When the **Duration** option is enabled on the **Inter-particle Collision Statistics** module (see also About Inter-particle Collision Statistics (p. 331)), this provides the standard deviation value of the duration of the collisions recorded by each individual whole particle or fragment, during an interval between two consecutive output times. **Note:** Does not apply to *meshed* (p. 1392) particles with particles upscaling enabled (see also About Meshed Particles Upscaling (p. 1033)). See also the "Collision statistics" section of the DEM Technical Manual (from the Rocky **Help** menu, point to **Manuals**, and then click **DEM Technical Manual**).
Force : Adhesive	When the **Force** option is enabled on the **Inter-particle Collision Statistics** module (see also About Inter-particle Collision Statistics (p. 331)), an **Adhesive Force** model (other than **None**) is enabled (see also About Physics Parameters (p. 417)), and an Adhesive **Force Fraction** for a particle-boundary and/or particle-particle Materials Interaction pair has a value set higher than zero (see also About Modifying Materials Interactions and Adhesion

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

Property Name	Description
	Values (p. 597)), this this provides the sum of the adhesive forces in the normal direction as recorded by each individual whole particle or fragment.
Force : Normal	When the **Force** option is enabled on the **Inter-particle Collision Statistics** module (see also About Inter-particle Collision Statistics (p. 331)), this provides the sum of the normal forces recorded by each individual whole particle or fragment.
Force : Tangential	When the **Force** option is enabled on the **Inter-particle Collision Statistics** module (see also About Inter-particle Collision Statistics (p. 331)), this provides the sum of the tangential forces recorded by each individual whole particle or fragment.
Frequency	When the **Frequency** option is enabled on the **Inter-particle Collision Statistics** module (see also About Inter-particle Collision Statistics (p. 331)), this provides the average collision frequency recorded by each individual whole particle or fragment, during an interval between two consecutive output times. See also the "Collision statistics" section of the DEM Technical Manual (from the Rocky **Help** menu, point to **Manuals**, and then click **DEM Technical Manual**).
Power : Dissipation	When the **Power** option is enabled on the **Inter-particle Collision Statistics** module (see also About Inter-particle Collision Statistics (p. 331)), this provides the power dissipated in the collisions of the contact forces recorded by each individual whole particle or fragment.
Power : Impact	When the **Power** option is enabled on the **Inter-particle Collision Statistics** module (see also About Inter-particle Collision Statistics (p. 331)), this provides the power transferred during the loading phase of all collisions recorded by each individual whole particle or fragment.
Power : Shear	When the **Power** option is enabled on the **Inter-particle Collision Statistics** module (see also About Inter-particle Collision Statistics (p. 331)), this provides the shear power of all collisions recorded by each individual whole particle or fragment.
Velocity : Impact : Normal : Kurtosis	When the **Normal Impact Velocity** option is enabled on the **Inter-particle Collision Statistics** module (see also About Inter-particle Collision Statistics (p. 331)), this provides the kurtosis value of the impact relative velocity in the normal direction resulting from the collisions recorded by each individual whole particle or fragment, during an interval between two consecutive output times. The kurtosis is a measure of how tall and sharp the central peak of a statistical distribution is. **Note:** Does not apply to *meshed* (p. 1392) particles with particles upscaling enabled (see also About Meshed Particles Upscaling (p. 1033)).

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

Property Name	Description
	See also the "Collision statistics" section of the DEM Technical Manual (from the Rocky **Help** menu, point to **Manuals**, and then click **DEM Technical Manual**).
Velocity : Impact : Normal : Mean	When the **Normal Impact Velocity** option is enabled on the **Inter-particle Collision Statistics** module (see also About Inter-particle Collision Statistics (p. 331)), this provides the mean value of the impact relative velocity in the normal direction resulting from the collisions recorded by each individual whole particle or fragment, during an interval between two consecutive output times. **Note:** Does not apply to *meshed* (p. 1392) particles with particles upscaling enabled (see also About Meshed Particles Upscaling (p. 1033)). See also the "Collision statistics" section of the DEM Technical Manual (from the Rocky **Help** menu, point to **Manuals**, and then click **DEM Technical Manual**).
Velocity : Impact : Normal : Skewness	When the **Normal Impact Velocity** option is enabled on the **Inter-particle Collision Statistics** module (see also About Inter-particle Collision Statistics (p. 331)), this provides the skewness value of the impact relative velocity in the normal direction resulting from the collisions recorded by each individual whole particle or fragment, during an interval between two consecutive output times. The skewness is a measure of the asymmetry of a statistical distribution about its mean. **Note:** Does not apply to *meshed* (p. 1392) particles with particles upscaling enabled (see also About Meshed Particles Upscaling (p. 1033)). See also the "Collision statistics" section of the DEM Technical Manual (from the Rocky **Help** menu, point to **Manuals**, and then click **DEM Technical Manual**).
Velocity : Impact : Normal : Std. Dev.	When the **Normal Impact Velocity** option is enabled on the **Inter-particle Collision Statistics** module (see also About Inter-particle Collision Statistics (p. 331)), this provides the standard deviation value of the impact relative velocity in the normal direction resulting from the collisions recorded by each individual whole particle or fragment, during an interval between two consecutive output times. **Note:** Does not apply to *meshed* (p. 1392) particles with particles upscaling enabled (see also About Meshed Particles Upscaling (p. 1033)).

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

Property Name	Description
	See also the "Collision statistics" section of the DEM Technical Manual (from the Rocky **Help** menu, point to **Manuals**, and then click **DEM Technical Manual**).
Velocity : Impact : Tangential : Kurtosis	When the **Tangential Impact Velocity** option is enabled on the **Inter-particle Collision Statistics** module (see also About Inter-particle Collision Statistics (p. 331)), this provides the kurtosis value of the impact relative velocity in the tangential direction resulting from the collisions recorded by each individual whole particle or fragment, during an interval between two consecutive output times. The kurtosis is a measure of how tall and sharp the central peak of a statistical distribution is. **Note:** Does not apply to *meshed* (p. 1392) particles with particles upscaling enabled (see also About Meshed Particles Upscaling (p. 1033)). See also the "Collision statistics" section of the DEM Technical Manual (from the Rocky **Help** menu, point to **Manuals**, and then click **DEM Technical Manual**).
Velocity : Impact : Tangential : Mean	When the **Tangential Impact Velocity** option is enabled on the **Inter-particle Collision Statistics** module (see also About Inter-particle Collision Statistics (p. 331)), this provides the mean value of the impact relative velocity in the tangential direction resulting from the collisions recorded by each individual whole particle or fragment, in between two output times. **Note:** Does not apply to *meshed* (p. 1392) particles with particles upscaling enabled (see also About Meshed Particles Upscaling (p. 1033)). See also the "Collision statistics" section of the DEM Technical Manual (from the Rocky **Help** menu, point to **Manuals**, and then click **DEM Technical Manual**).
Velocity : Impact : Tangential : Skewness	When the **Tangential Impact Velocity** option is enabled on the **Inter-particle Collision Statistics** module (see also About Inter-particle Collision Statistics (p. 331)), this provides the skewness value of the impact relative velocity in the tangential direction resulting from the collisions recorded by each individual whole particle or fragment, during an interval between two consecutive output times. The skewness is a measure of the asymmetry of a statistical distribution about its mean. **Note:** Does not apply to *meshed* (p. 1392) particles with particles upscaling enabled (see also About Meshed Particles Upscaling (p. 1033)).

Property Name	Description
	See also the "Collision statistics" section of the DEM Technical Manual (from the Rocky **Help** menu, point to **Manuals**, and then click **DEM Technical Manual**).
Velocity : Impact : Tangential : Std. Dev.	When the **Tangential Impact Velocity** option is enabled on the **Inter-particle Collision Statistics** module (see also About Inter-particle Collision Statistics (p. 331)), this provides the standard deviation value of the impact relative velocity in the tangential direction resulting from the collisions recorded by each individual whole particle or fragment, during an interval between two consecutive output times. **Note:** Does not apply to *meshed* (p. 1392) particles with particles upscaling enabled (see also About Meshed Particles Upscaling (p. 1033)). See also the "Collision statistics" section of the DEM Technical Manual (from the Rocky **Help** menu, point to **Manuals**, and then click **DEM Technical Manual**).
Particle Instantaneous Energies Properties	
Energy : Kinetic : Rotational	When the **Particle Instantaneous Energies** module is enabled (see also About Particle Instantaneous Energies (p. 341)), this is the amount of rotational kinetic energy measured for each individual whole particle or fragment. Rotational kinetic energy is the energy due to the rotation of a particle around an axis through its center of mass. It is given by: $E^{k,r}=\frac{1}{2}\omega_p^T I \omega_p$ where: ω_p is the particle's angular velocity, and I is the moment of inertia tensor.
Energy : Kinetic : Translational	When the **Particle Instantaneous Energies** module is enabled (see also About Particle Instantaneous Energies (p. 341)), this is the amount of translational kinetic energy measured for each individual whole particle or fragment. Translational kinetic energy is the energy related to the rectilinear motion of a particle. It is given by: $E^{k,t}=\frac{1}{2}m_p \mathrm{v}_p^2$ where: m_p is the particle's mass, and v_p is the velocity of its center of mass.
Energy : Potential	When the **Particle Instantaneous Energies** module is enabled (see also About Particle Instantaneous Energies (p. 341)), this is the amount of potential energy measured for each individual whole particle or fragment. Potential energy is the energy of a particle due to its position relative to the Earth's gravitational field. In Rocky it is calculated as: $E^p=m_p \mathrm{g}\cdot \mathrm{r}_p$ where: m_p is the particle's mass, g is the acceleration due to gravity, and r_p is the position vector of the particle's center of mass. Conventionally, a zero potential energy is attributed to a plane orthogonal to the gravity direction, passing through the origin of the coordinate system.

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

Property Name	Description
Coarse Grain Modeling Properties	
Particle CGM Scale Factor	When **Coarse Grain Modeling** is enabled (see also About Physics Parameters (p. 417)), this is the **CGM Scale Factor** that was applied to the particle **Size** (**Diameter** or **Scale Factor**) (see also About Adding and Editing Particle Sets (p. 602)) and which determined the final simulated size of the particle and how many smaller particles' behavior it represents.
CFD Coupling Particle Statistics Properties	
Heat : Convective : Transfer Rate	When the **Convective Heat Transfer Rate** option is enabled on the **CFD Coupling Particle Statistics** module (see also About Fluid-Related Statistics for Particles (p. 335)), this provides the average convective heat transfer rate exchanged between a particle and the surrounding fluids as recorded by each individual whole particle or fragment, during an interval between two consecutive *output time* (p. 1393). **Note:** Data is provided only when the **Thermal Model** is enabled and a **Convective Heat Transfer Law** has been defined for the fluid-particle interactions.
Force : Drag	When the **Drag Force** option is enabled on the **CFD Coupling Particle Statistics** module (see also About Fluid-Related Statistics for Particles (p. 335)), this provides the average drag force induced by the fluid flow as recorded by each individual whole particle or fragment, during an interval between two consecutive output times.
Torque : Flow-Induced	When the **Flow-Induced Torque** option is enabled on the **CFD Coupling Particle Statistics** module (see also About Fluid-Related Statistics for Particles (p. 335)), this provides the average torque induced by the fluid flow as recorded by each individual whole particle or fragment, during an interval between two consecutive output times. **Note:** Data is collected only when a **Torque Law** has been defined for the fluid-particle interactions.
Force : Lift	When the **Lift Force** option is enabled on the **CFD Coupling Particle Statistics** module (see also About Fluid-Related Statistics for Particles (p. 335)), this provides the average lift force induced by the fluid flow as recorded by each

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

Property Name	Description
	individual whole particle or fragment, during an interval between two consecutive output times. **Note:** Data is collected only when a **Lift Law** has been defined for the fluid-particle interactions.
Force : Pressure Gradient	When the **Pressure Gradient Force** option is enabled on the **CFD Coupling Particle Statistics** module (see also About Fluid-Related Statistics for Particles (p. 335)), this provides the average pressure gradient force induced by the fluid flow as recorded by each individual whole particle or fragment, during an interval between two consecutive output times.
Force : Virtual Mass	When the **Virtual Mass Force** option is enabled on the **CFD Coupling Particle Statistics** module (see also About Fluid-Related Statistics for Particles (p. 335)), this provides the average virtual mass force induced by the fluid flow as recorded by each individual whole particle or fragment, during an interval between two consecutive output times. **Note:** Data is collected only when a **Virtual Mass Law** has been defined for the fluid-particle interactions.

PARTICLE SET PROPERTIES (PER SET ONLY)

These properties are available from an individual Particle set (p. 1393), but are collected and available only when the Intra-particle Collision Statistics feature is enabled and the particle set is a type that supports the collection of that data (see also About Intra-Particle Collision Statistics (p. 324)).

Note:

For all Intra-particle Collisions Statistics properties, the effects of upscaling will be ignored for any *meshed* (p. 1392) particles with **Meshed Particles Upscaling** enabled (see also About Meshed Particles Upscaling (p. 1033)).

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

Figure 5.9: Supported Particle set, Properties options on the Data Editors panel when Intra-particle Collision Statistics are collected

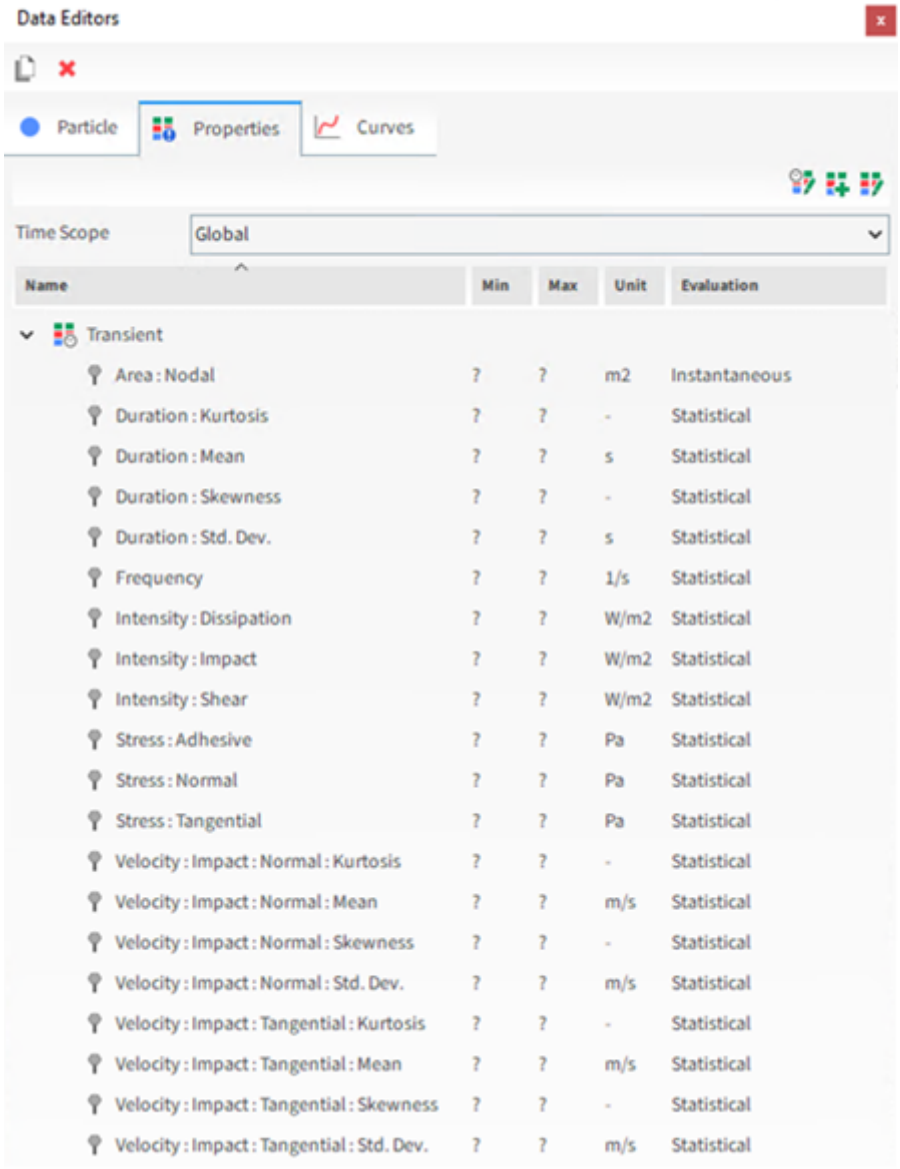


Table 5: Properties available for an individual Particle set that supports the collection of Intra-particle Collision Statistics

Property Name	Description
Area : Nodal	When the **Intra-particle Collisions Statistic** module is enabled (see also About Intra-Particle Collision Statistics (p. 324)), this provides the surface area making up the different regions of influence surrounding the vertices of the representative particle of the selected particle set. This is useful for defining custom properties, such as frequency / area, intensity area, and so on. See also the "Collision statistics" section of the DEM Technical Manual (from the Rocky **Help** menu, point to **Manuals**, and then click **DEM Technical Manual**).

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

Property Name	Description
Duration : Kurtosis	When the **Duration** option is enabled on the **Intra-particle Collision Statistics** module (see also About Intra-Particle Collision Statistics (p. 324)), this provides the kurtosis value of the duration of the collisions recorded in different regions of the representative particle of the selected particle set, during an interval between two consecutive *output time* (p. 1393). The kurtosis is a measure of how tall and sharp the central peak of a statistical distribution is. See also the "Collision statistics" section of the DEM Technical Manual (from the Rocky **Help** menu, point to **Manuals**, and then click **DEM Technical Manual**).
Duration : Mean	When the **Duration** option is enabled on the **Intra-particle Collision Statistics** module (see also About Intra-Particle Collision Statistics (p. 324)), this provides the mean value of the duration of the collisions recorded in different regions of the representative particle of the selected particle set, during an interval between two consecutive output times. See also the "Collision statistics" section of the DEM Technical Manual (from the Rocky **Help** menu, point to **Manuals**, and then click **DEM Technical Manual**).
Duration : Skewness	When the **Duration** option is enabled on the **Intra-particle Collision Statistics** module (see also About Intra-Particle Collision Statistics (p. 324)), this provides the skewness value of the duration of the collisions recorded in different regions of the representative particle of the selected particle set, during an interval between two consecutive output times. The skewness is a measure of the asymmetry of a statistical distribution about its mean. See also the "Collision statistics" section of the DEM Technical Manual (from the Rocky **Help** menu, point to **Manuals**, and then click **DEM Technical Manual**).
Duration : Std. Dev.	When the **Duration** option is enabled on the **Intra-particle Collision Statistics** module (see also About Intra-Particle Collision Statistics (p. 324)), this provides the standard deviation value of the duration of the collisions recorded in different regions of the representative particle of the selected particle set, during an interval between two consecutive output times. See also the "Collision statistics" section of the DEM Technical Manual (from the Rocky **Help** menu, point to **Manuals**, and then click **DEM Technical Manual**).
Frequency	When the **Frequency** option is enabled on the **Intra-particle Collision Statistics** module (see also About Intra-Particle Collision Statistics (p. 324)), this provides the average collision frequency per particle recorded in different regions of the representative particle of the selected particle set, during an interval between two consecutive output times. See also the "Collision statistics" section of the DEM Technical Manual (from the Rocky **Help** menu, point to **Manuals**, and then click **DEM Technical Manual**).
Intensity : Dissipation	When the **Intensities** option is enabled on the **Intra-particle Collision Statistics** module (see also About Intra-Particle Collision Statistics (p. 324)), this provides the average power dissipated per unit area by the contact forces resulting from the collisions recorded in different regions of the representative particle of the selected particle set, during an interval between two consecutive output times.

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

Property Name	Description
	See also the "Collision statistics" section of the DEM Technical Manual (from the Rocky **Help** menu, point to **Manuals**, and then click **DEM Technical Manual**).
Intensity : Impact	When the **Intensities** option is enabled on the **Intra-particle Collision Statistics** module (see also About Intra-Particle Collision Statistics (p. 324)), this provides the average impact power divided by the area made by the contact forces resulting from the collisions recorded in different regions of the representative particle of the selected particle set, during an interval between two consecutive output times.
Intensity : Shear	When the **Intensities** option is enabled on the **Intra-particle Collision Statistics** module (see also About Intra-Particle Collision Statistics (p. 324)), this provides the average shear power made by the contact forces resulting from the collisions, the average shear power by area recorded in different regions of the representative particle of the selected particle set, during an interval between two consecutive output times.
Stress : Adhesive	When the **Stresses** option is enabled on the **Intra-particle Collision Statistics** module (see also About Intra-Particle Collision Statistics (p. 324)), an **Adhesive Force model (other than None**) is enabled (see also About Physics Parameters (p. 417)), and an Adhesive **Force Fraction** for a particle-boundary and/or particle-particle Materials Interaction pair has a value set higher than zero (see also About Modifying Materials Interactions and Adhesion Values (p. 597)), this provides the average adhesive stress resulting from the collisions recorded in different regions of the representative particle of the selected particle set, during an interval between two consecutive output times. See also the "Collision statistics" section of the DEM Technical Manual (from the Rocky **Help** menu, point to **Manuals**, and then click **DEM Technical Manual**).
Stress : Normal	When the **Stresses** option is enabled on the **Intra-particle Collision Statistics** module (see also About Intra-Particle Collision Statistics (p. 324)), this provides the average normal stress resulting from the collisions recorded in different regions of the representative particle of the selected particle set, during an interval between two consecutive output times. See also the "Collision statistics" section of the DEM Technical Manual (from the Rocky **Help** menu, point to **Manuals**, and then click **DEM Technical Manual**).
Stress : Tangential	When the **Stresses** option is enabled on the **Intra-particle Collision Statistics** module (see also About Intra-Particle Collision Statistics (p. 324)), this provides the average tangential stress resulting from the collisions recorded in different regions of the representative particle of the selected particle set, during an interval between two consecutive output times. See also the "Collision statistics" section of the DEM Technical Manual (from the Rocky **Help** menu, point to **Manuals**, and then click **DEM Technical Manual**).
Velocity : Impact : Normal : Kurtosis	When the **Normal Impact Velocity** option is enabled on the **Intra-particle Collision Statistics** module (see also About Intra-Particle Collision Statistics (p. 324)), this provides the kurtosis value of the impact relative velocity in the normal direction resulting from the collisions recorded in different regions of the representative particle of the selected particle set, during an interval

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

Property Name	Description
	between two consecutive output times. The kurtosis is a measure of how tall and sharp the central peak of a statistical distribution is. See also the "Collision statistics" section of the DEM Technical Manual (from the Rocky **Help** menu, point to **Manuals**, and then click **DEM Technical Manual**).
Velocity : Impact : Normal : Mean	When the **Normal Impact Velocity** option is enabled on the **Intra-particle Collision Statistics** module (see also About Intra-Particle Collision Statistics (p. 324)), this provides the mean value of the impact relative velocity in the normal direction resulting from the collisions recorded in different regions of the representative particle of the selected particle set, during an interval between two consecutive output times. See also the "Collision statistics" section of the DEM Technical Manual (from the Rocky **Help** menu, point to **Manuals**, and then click **DEM Technical Manual**).
Velocity : Impact : Normal : Skewness	When the **Normal Impact Velocity** option is enabled on the **Intra-particle Collision Statistics** module (see also About Intra-Particle Collision Statistics (p. 324)), this provides the skewness value of the impact relative velocity in the normal direction resulting from the collisions recorded in different regions of the representative particle of the selected particle set, during an interval between two consecutive output times. The skewness is a measure of the asymmetry of a statistical distribution about its mean. See also the "Collision statistics" section of the DEM Technical Manual (from the Rocky **Help** menu, point to **Manuals**, and then click **DEM Technical Manual**).
Velocity : Impact : Normal : Std. Dev.	When the **Normal Impact Velocity** option is enabled on the **Intra-particle Collision Statistics** module (see also About Intra-Particle Collision Statistics (p. 324)), this provides the standard deviation value of the impact relative velocity in the normal direction resulting from the collisions recorded in different regions of the representative particle of the selected particle set, during an interval between two consecutive output times. See also the "Collision statistics" section of the DEM Technical Manual (from the Rocky **Help** menu, point to **Manuals**, and then click **DEM Technical Manual**).
Velocity : Impact : Tangential : Kurtosis	When the **Tangential Impact Velocity** option is enabled on the **Intra-particle Collision Statistics** module (see also About Intra-Particle Collision Statistics (p. 324)), this provides the kurtosis value of the impact relative velocity in the tangential direction resulting from the collisions recorded in different regions of the representative particle of the selected particle set, during an interval between two consecutive output times. The kurtosis is a measure of how tall and sharp the central peak of a statistical distribution is. See also the "Collision statistics" section of the DEM Technical Manual (from the Rocky **Help** menu, point to **Manuals**, and then click **DEM Technical Manual**).
Velocity : Impact : Tangential : Mean	When the **Tangential Impact Velocity** option is enabled on the **Intra-particle Collision Statistics** module (see also About Intra-Particle Collision Statistics (p. 324)), this provides the mean value of the impact relative velocity in the tangential direction resulting from the collisions recorded in different

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

Property Name	Description
	regions of the representative particle of the selected particle set, during an interval between two consecutive output times. See also the "Collision statistics" section of the DEM Technical Manual (from the Rocky **Help** menu, point to **Manuals**, and then click **DEM Technical Manual**).
Velocity : Impact : Tangential : Skewness	When the **Tangential Impact Velocity** option is enabled on the **Intra-particle Collision Statistics** module (see also About Intra-Particle Collision Statistics (p. 324)), this provides the skewness value of the impact relative velocity in the tangential direction resulting from the collisions recorded in different regions of the representative particle of the selected particle set, during an interval between two consecutive output times. The skewness is a measure of the asymmetry of a statistical distribution about its mean. See also the "Collision statistics" section of the DEM Technical Manual (from the Rocky **Help** menu, point to **Manuals**, and then click **DEM Technical Manual**).
Velocity : Impact : Tangential : Std. Dev.	When the **Tangential Impact Velocity** option is enabled on the **Intra-particle Collision Statistics** module (see also About Intra-Particle Collision Statistics (p. 324)), this provides the standard deviation value of the impact relative velocity in the tangential direction resulting from the collisions recorded in different regions of the representative particle of the selected particle set, during an interval between two consecutive output times. See also the "Collision statistics" section of the DEM Technical Manual (from the Rocky **Help** menu, point to **Manuals**, and then click **DEM Technical Manual**).

CONTACTS PROPERTIES

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

Figure 5.10: Contacts Properties options on the Data Editors panel

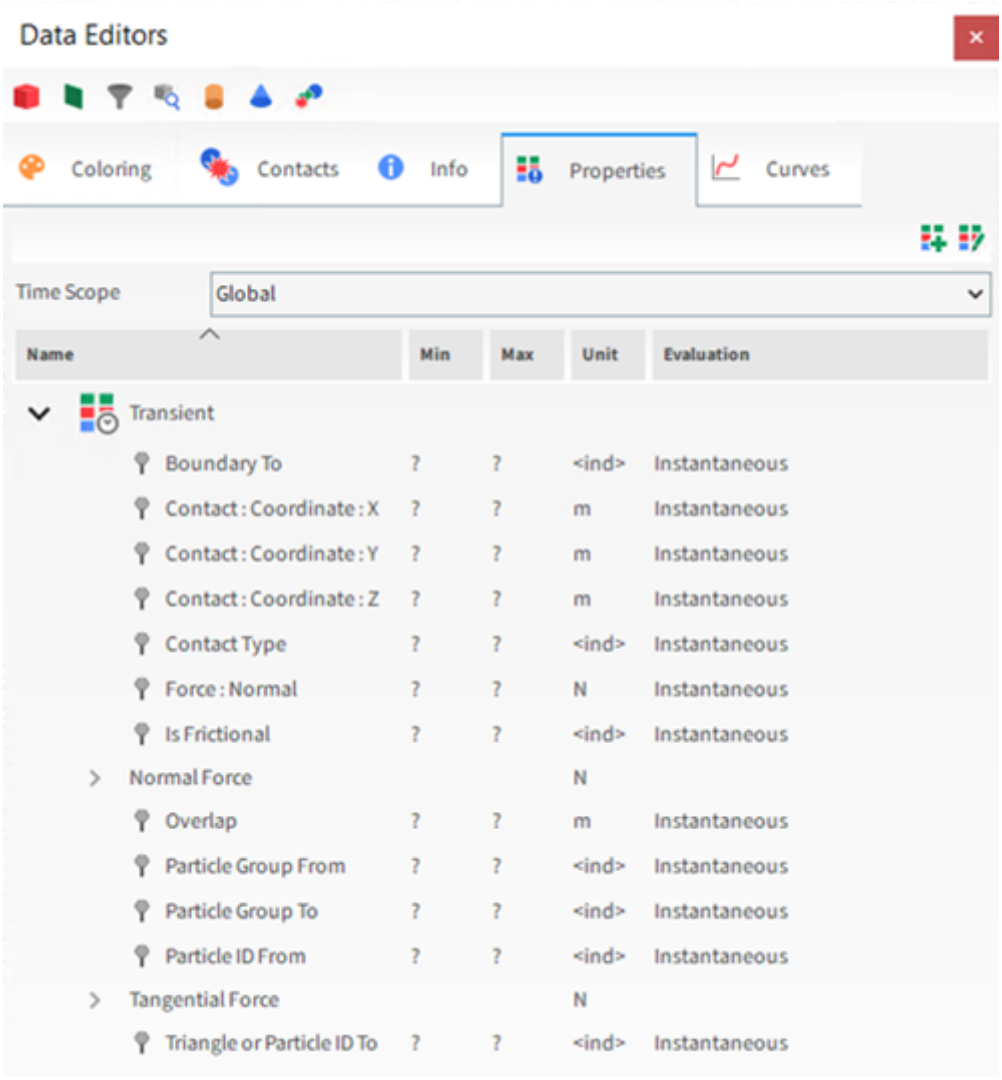


Table 6: Properties available by default when Contacts data is collected

Property Name	Description
Absolute Normal Force	The current normal force magnitude of the contact.
Boundary To	Identifies the parent entity of the object being contacted and for boundaries specifically, identifies the specific component. No matter whether they have sustained contacts or not, the list always includes all geometry components and inlets, numbered from 0 (zero) in (roughly) the order in which they were added. It also includes the main Particle

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

Property Name	Description
	category (representing all *Particle Group* (p. 1393)), which will always be displayed as -1. **Tip:** The numbers are needed, in part, when creating a Property User Process (see also About the Property User Process (p. 1202)). Use the Color scale to map the numbers to the object name (see also About Color Scales (p. 1086)).
Contact Type	The type of contact being made. Specifically: • **Particle -> Boundary Contact** indicates that a particle is contacting a geometry component. • **Particle -> Particle Contact** indicates that a particle is contacting another particle.
Contact : Coordinate : X	The coordinate location of the contact along the X-axis.
Contact : Coordinate : Y	The coordinate location of the contact along the Y-axis.
Contact : Coordinate : Z	The coordinate location of the contact along the Z-axis.
Initial Contact Overlap	The initial magnitude of the contact overlap between the contact pair (particle-particle or particle boundary).
Is Frictional	Indicates whether the contact is frictional and adhesive, or only adhesive. Specifically: • A **1** indicates a frictional and adhesive contact. • A **0** indicates an adhesive-only contact.
Force : Normal : X	The current normal force of the contact in the X direction.
Force : Normal : Y	The current normal force of the contact in the Y direction.
Force : Normal : Z	The current normal force of the contact in the Z direction.
Overlap	The current magnitude of the contact overlap between the contact pair (particle-particle or particle boundary).
Particle Group From	Displays the name of the *Particle Group* (p. 1393) doing the contacting. **Note:** In Rocky, the object doing the contact is always a particle. Even if a boundary in motion collides with a particle, Rocky still considers the particle as the originator of the contact.

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

Property Name	Description
Particle Group To	Identifies the parent entity of the object being contacted and for Particles specifically, identifies the specific *Particle Group* (p. 1393) being contacted. No matter whether they have sustained contacts or not, the list always includes all Particle sets, numbered from 0 (zero) in (roughly) the order in which they were added. It also includes the main Boundary category (representing all geometry components and inlets), which will always be displayed as -1. **Tip:** The numbers are needed, in part, when creating a Property User Process (see also About the Property User Process (p. 1202)). Use the Color scale to map the numbers to the object name (see also About Color Scales (p. 1086)).
Particle ID From	Displays the unique identifier (particle ID) associated with the particle doing the contacting. **Note:** In Rocky, the object doing the contact is always a particle. Even if a boundary in motion collides with a particle, Rocky still considers the particle as the originator of the contact.
Force : Tangential : X	The current tangential force of the contact in the X direction.
Force : Tangential : Y	The current tangential force of the contact in the Y direction.
Force : Tangential : Z	The current tangential force of the contact in the Z direction.
Triangle or Particle ID To	Displays the unique identifier (ID) associated with the *cell* (p. 1389) being contacted. Specifically: • For Geometries, this is the triangle ID. • For Particles, this is the particle ID.

SPH PROPERTIES

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

Figure 5.11: SPH Properties options on the Data Editors panel

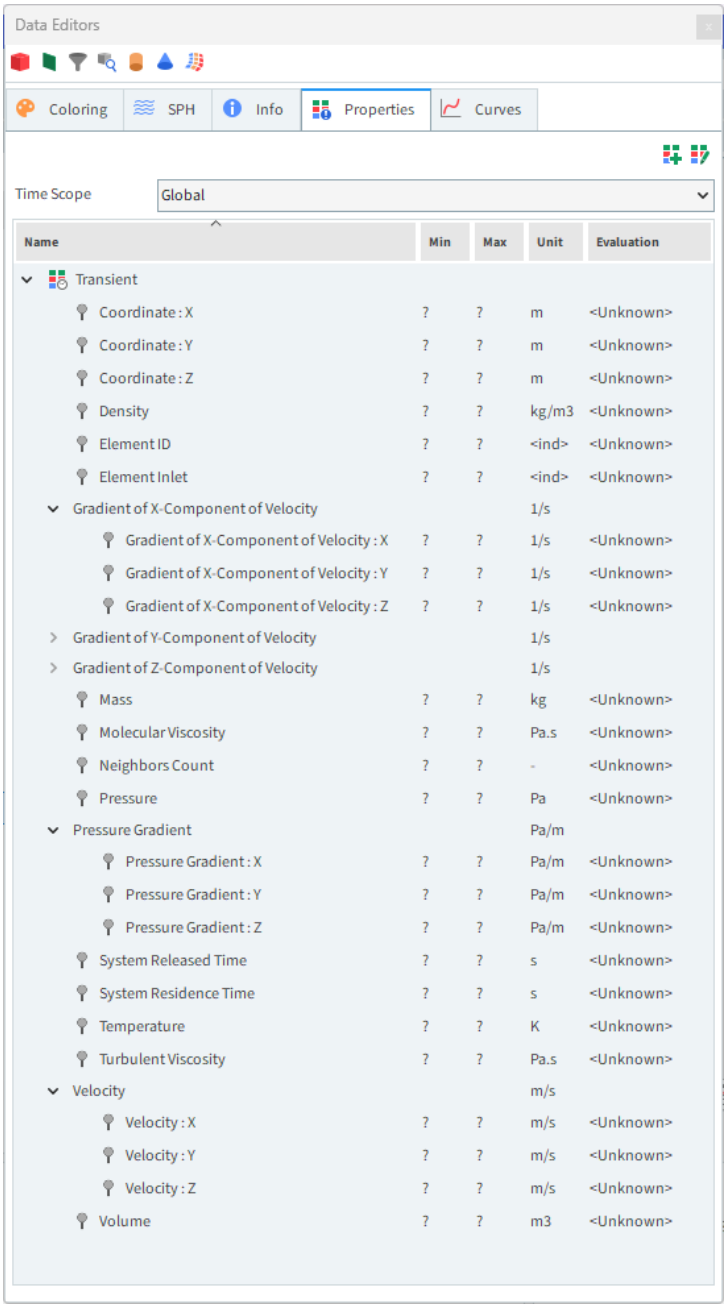


Table 7: Properties available when running a SPH simulation

Property Name	Description
Standard Properties	
Coordinate : X	The Coordinate component in the X direction for each SPH element.
Coordinate : Y	The Coordinate component in the Y direction for each SPH element.
Coordinate : Z	The Coordinate component in the Z direction for each SPH element.

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

Property Name	Description
Density	The density of each SPH element
Element ID	The numeric identifier unique to each individual SPH element.
Element Inlet	The name of the inlet through which each individual SPH element originally entered the simulation.
Gradient of X-Component of Velocity : X	The X-direction gradient of the X-component of Velocity.
Gradient of X-Component of Velocity : Y	The Y-direction gradient of the X-component of Velocity.
Gradient of X-Component of Velocity : Z	The Z-direction gradient of the X-component of Velocity.
Gradient of Y-Component of Velocity : X	The X-direction gradient of the Y-component of Velocity.
Gradient of Y-Component of Velocity : Y	The Y-direction gradient of the Y-component of Velocity.
Gradient of Y-Component of Velocity : Z	The Z-direction gradient of the Y-component of Velocity.
Gradient of Z-Component of Velocity : X	The X-direction gradient of the Z-component of Velocity.
Gradient of Z-Component of Velocity : Y	The Y-direction gradient of the Z-component of Velocity.
Gradient of Z-Component of Velocity : Z	The Z-direction gradient of the Z-component of Velocity.
Mass	The mass of each individual SPH element.
Molecular Viscosity	The viscosity associated to each SPH element.
Neighbors Count	Count of neighbor SPH elements.
Pressure	The pressure that each SPH element is subjected to.
Pressure Gradient : X	The Pressure gradient in the X-direction.
Pressure Gradient : Y	The Pressure gradient in the Y-direction.
Pressure Gradient : Z	The Pressure gradient in the Z-direction.
System Released Time	The instance in simulation time when each individual SPH element originally entered the simulation domain.
System Residence Time	The amount of time each individual SPH element has been within the simulation domain. It can be defined as the difference between the current time, t, and the time the SPH element entered the simulation, t_{in}. Therefore, $r_{local}=t-t_{in}$.
Temperature	Temperature of SPH elements.

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

Property Name	Description
Turbulent Viscosity	The turbulent viscosity associated to each SPH element. **Note:** This property is only available if a turbulence model is enabled.
Velocity : X	The velocity vector component in the X direction for each SPH element.
Velocity : Y	The velocity vector component in the Y direction for each SPH element.
Velocity : Z	The velocity vector component in the Z direction for each SPH element.
Volume	The volume of each individual SPH element.

(See also About Contacts (p. 746)).

1-WAY LBM PROPERTIES

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

Figure 5.12: 1-Way LBM, Properties options on the Data Editors panel

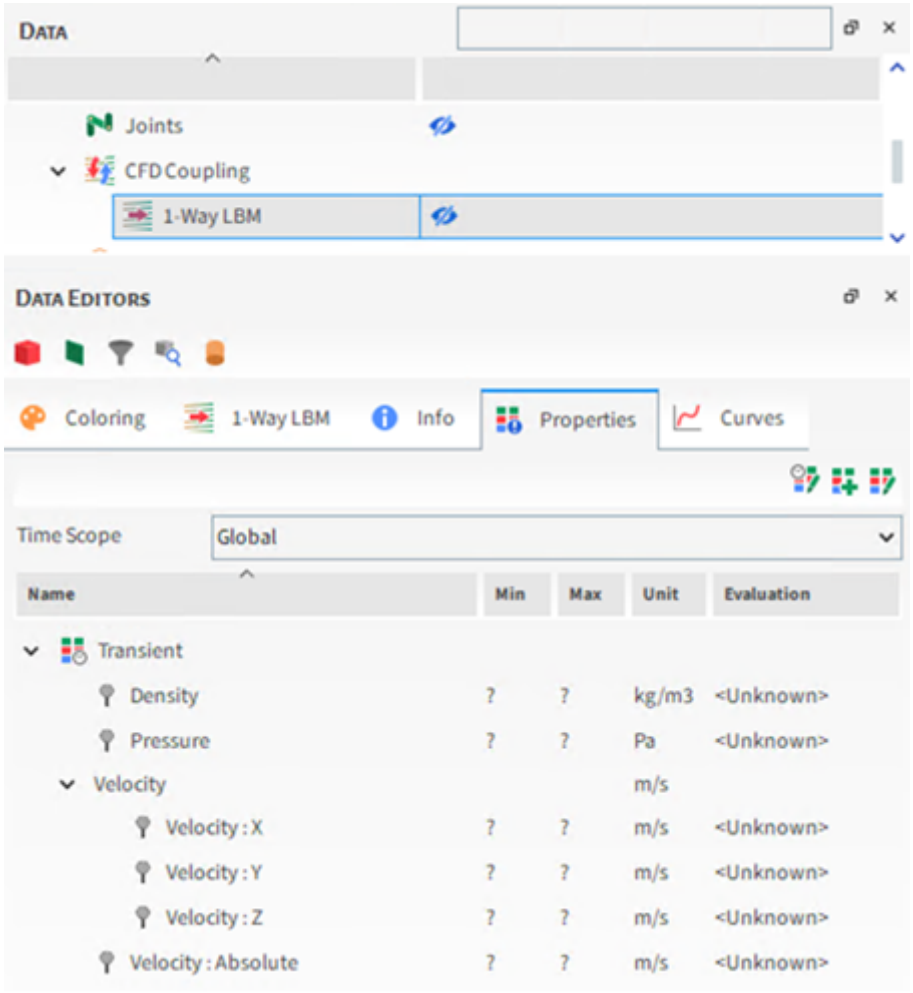


Table 8: Properties available by default for 1-Way LBM (CFD Coupling)

Property Name	Description
Density	The density the fluid.
Pressure	The pressure at each individual fluid node.
Velocity	The velocity vector of each individual fluid node, the magnitude of which is made up of the x, y, and z components respectively. **Note:** Because this is a vector, display is limited to the 3D View only.
Velocity : X	The x-component of the fluid velocity in global coordinates.
Velocity : Y	The y-component of the fluid velocity in global coordinates.
Velocity : Z	The z-component of the fluid velocity in global coordinates.

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

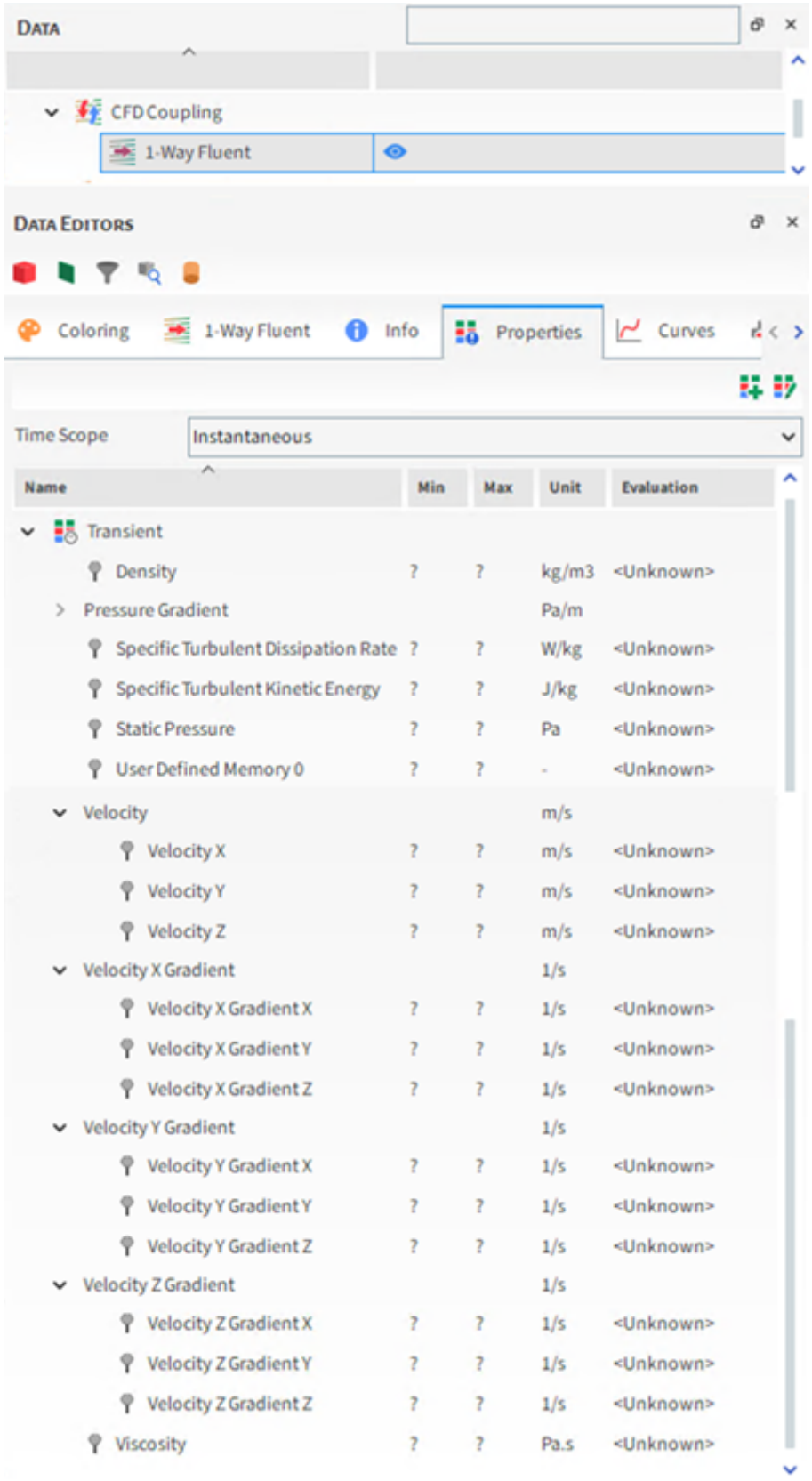
Property Name	Description
Velocity : Absolute	The absolute velocity of each individual fluid node.

(See also About Using the 1-Way LBM Method (p. 771).)

1-WAY FLUENT PROPERTIES

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

Figure 5.13: Fluent 1-Way Instantaneous, Properties options in the Data Editors panel



Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

Figure 5.14: Fluent 1-Way variables are selected during the export process

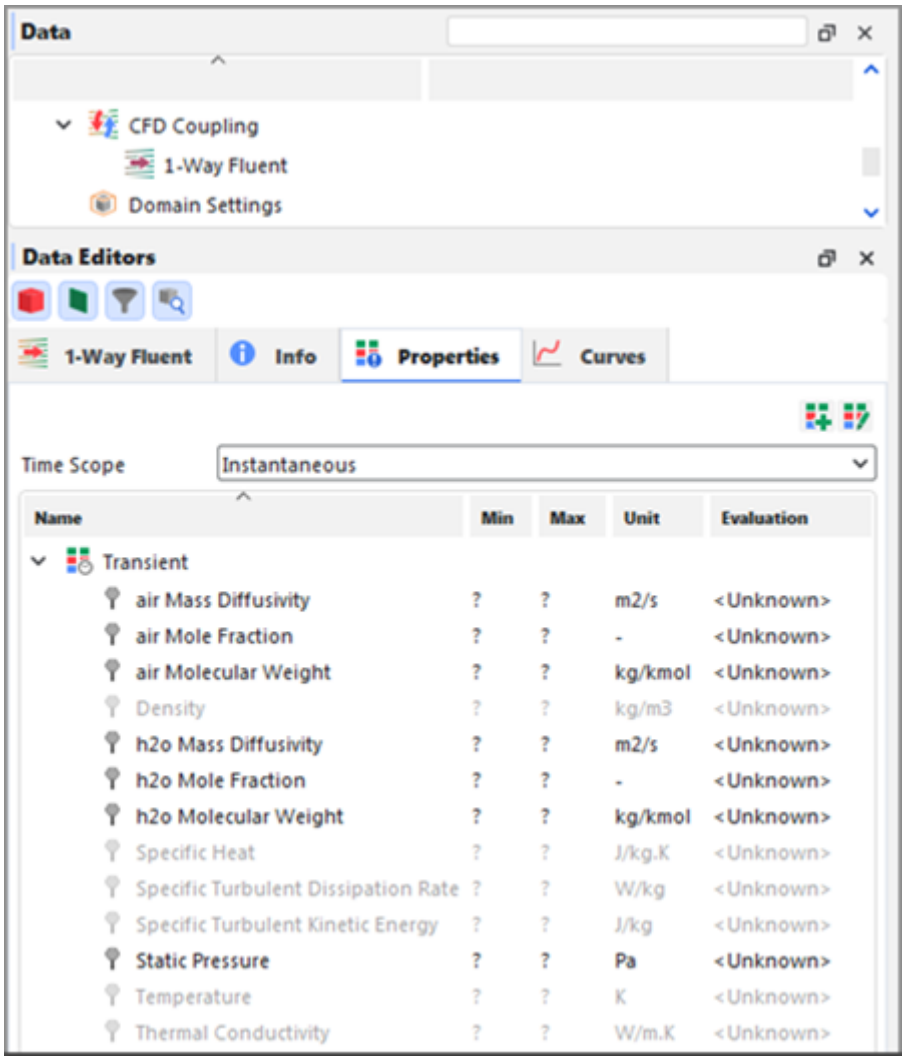


Table 9: Properties available by default for 1-Way Fluent (CFD Coupling)

Property Name	Description
Density	The density of the fluid. For *multiphase* (p. 1392) flows, the mixture density is reported.
Pressure	The static pressure of the fluid.
Pressure Gradient	The pressure gradient of the fluid, the magnitude of which is made up of the x, y, and z components respectively.
Pressure Gradient X	The pressure gradient of the fluid in the X direction.
Pressure Gradient Y	The pressure gradient of the fluid in the Y direction.
Pressure Gradient Z	The pressure gradient of the fluid in the Z direction.
Specific Turbulent Dissipation Rate	When the .f2r file has a turbulent model that uses turbulence dissipation rate as one of its quantities, this is the specific turbulent dissipation rate of the fluid.

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

Property Name	Description
Specific Turbulent Kinetic Energy	When the .f2r file has a turbulent model that uses turbulence kinetic energy as one of its quantities, this is the specific turbulent kinetic energy of the fluid.
Specific Heat	When **Thermal Model** is enabled (see also Enable Thermal Modeling (p. 400)) and the Fluent case has a thermal solution, this is the specific heat of the fluid. For *multiphase* (p. 1392) flows, the specific heat of the mixture is reported.
Temperature	When **Thermal Model** is enabled (see also Enable Thermal Modeling (p. 400)) and the Fluent case has a thermal solution, this is the thermodynamic temperature of the fluid.
Thermal Conductivity	When **Thermal Model** is enabled (see also Enable Thermal Modeling (p. 400)) and the Fluent case has a thermal solution, this is the thermal conductivity of the fluid. For *multiphase* (p. 1392) flows, the thermal conductivity of the mixture is reported.
Velocity	The velocity vector of the fluid, the magnitude of which is made up of the x, y, and z components respectively. **Note:** Because this is a vector, display is limited to the 3D View only.
Velocity X	The velocity of the fluid along the X-axis.
Velocity Y	The velocity of the fluid along the Y-axis.
Velocity Z	The velocity of the fluid along the Z-axis.
Velocity X Gradient	The gradient of the velocity of the fluid along the X-axis.
Velocity X Gradient X	The gradient of the X Velocity in the X direction.
Velocity X Gradient Y	The gradient of the X Velocity in the Y direction.
Velocity X Gradient Z	The gradient of the X Velocity in the Z direction.
Velocity Y Gradient	The gradient of the velocity of the fluid along the Y-axis.
Velocity Y Gradient X	The gradient of the Y Velocity in the X direction.
Velocity Y Gradient Y	The gradient of the Y Velocity in the Y direction.
Velocity Y Gradient Z	The gradient of the Y Velocity in the Z direction.
Velocity Z Gradient	The gradient of the velocity of the fluid along the Z-axis.
Velocity Z Gradient X	The gradient of the Z Velocity in the X direction.
Velocity Z Gradient Y	The gradient of the Z Velocity in the Y direction.
Velocity Z Gradient Z	The gradient of the Z Velocity in the Z direction.
Viscosity	The viscosity of the fluid. For *multiphase* (p. 1392) flows, the viscosity of the mixture is reported.

(See also About Using the 1-Way Fluent Method (p. 775)).

2-WAY FLUENT PROPERTIES

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

Figure 5.15: 2-Way Fluent Properties in the Data Editors panel

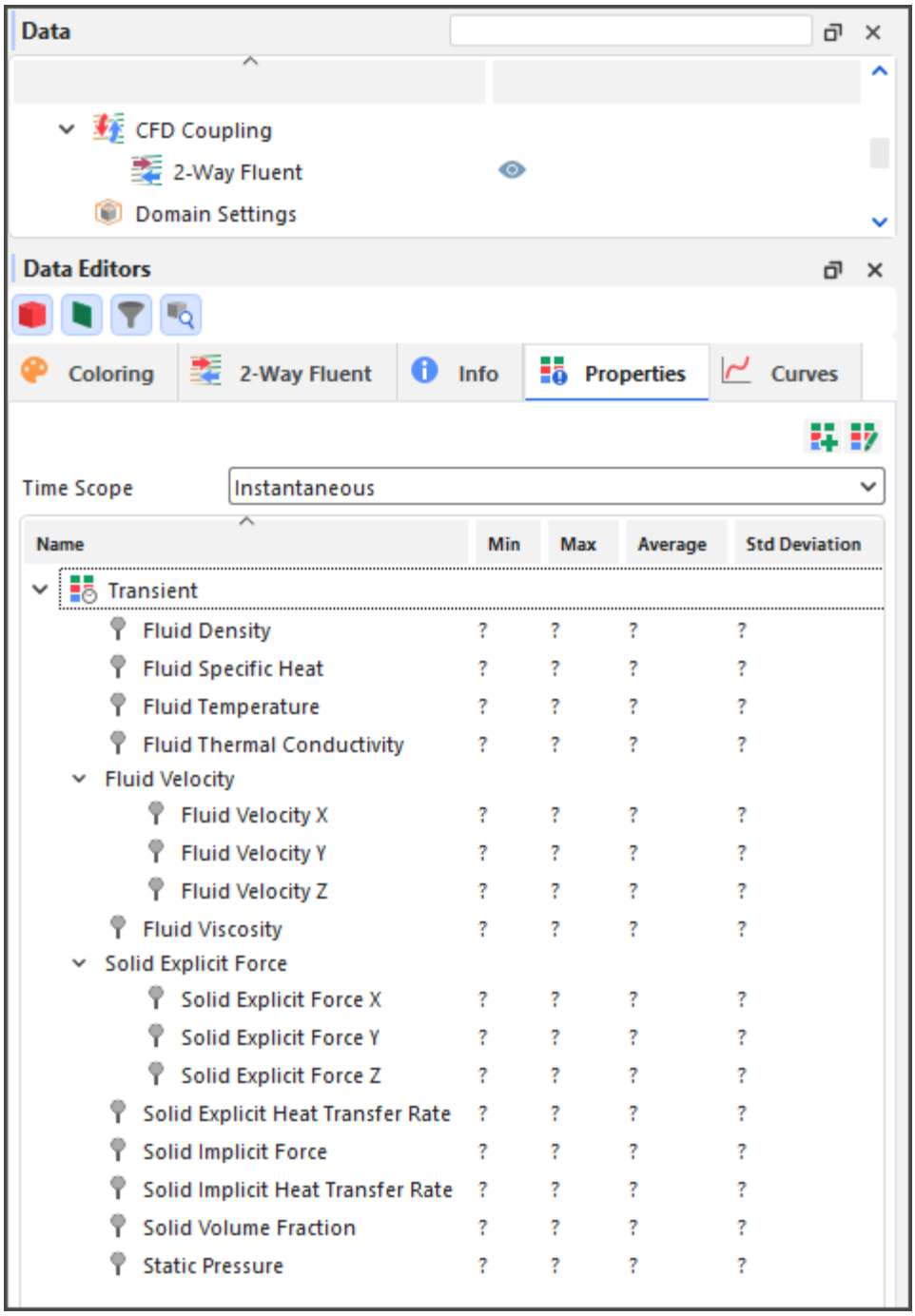


Table 10: Properties available by default for 2-Way Fluent

Property Name	Description
Fluid Density	The density of the fluid. For *multiphase* (p. 1392) flows, the density o the mixture is reported.
Fluid Specific Heat	When **Thermal Model** is enabled (see also Enable Thermal Modeling (p. 400)) and the Fluent case has a thermal solution, this is the specific heat of the fluid. For *multiphase* (p. 1392) flows, the specific heat of the mixture is reported.

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

Property Name	Description
Fluid Temperature	When **Thermal Model** is enabled (see also Enable Thermal Modeling (p. 400)) and the Fluent case has a thermal solution, this is the thermodynamic temperature of the fluid.
Fluid Thermal Conductivity	When **Thermal Model** is enabled (see also Enable Thermal Modeling (p. 400)) and the Fluent case has a thermal solution, this is the thermal conductivity of the fluid. For *multiphase* (p. 1392) flows, the thermal conductivity of the mixture is reported.
Fluid Turbulent Dissipation Rate	When the Fluent case has a turbulent model that uses turbulence dissipation as one of its quantities, this is the turbulent dissipation rate of the fluid.
Fluid Turbulent Kinetic Energy	When the Fluent case has a turbulent model that uses turbulence kinetic energy as one of its quantities, this is the turbulent kinetic energy of the fluid.
Fluid Velocity	The velocity vector of the fluid, the magnitude of which is made up of the x, y, and z components respectively. **Note:** Because this is a vector, display is limited to the 3D View only.
Fluid Velocity X	The velocity of the fluid in the x direction.
Fluid Velocity Y	The velocity of the fluid in the y direction.
Fluid Velocity Z	The velocity of the fluid in the z direction.
Fluid Viscosity	The viscosity of the fluid. For *multiphase* (p. 1392) flows, the viscosity of the mixture is reported.
Solid Explicit Force	The explicit portion of the total momentum contribution from particle-fluid interaction forces, which is in turn given by: $A+Bv$ where: A is the explicit contribution, B is the implicit contribution coefficient, and v is the fluid velocity. Notes: • This expression is valid for each (x,y,z) vector component. • Because this is a vector, display is limited to the 3D View only.
Solid Explicit Force X	The explicit momentum source added to the fluid momentum equation in the x direction.
Solid Explicit Force Y	The explicit momentum source added to the fluid momentum equation in the y direction.

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

Property Name	Description
Solid Explicit Force Z	The explicit momentum source added to the fluid momentum equation in the z direction.
Solid Explicit Heat Transfer Rate	When **Thermal Model** is enabled (see also Enable Thermal Modeling (p. 400)) and the Fluent case has a thermal solution, this is the explicit heat transfer rate source contribution of the particles into the fluid, where the total heat source is given by: $A+BT$ where: A is the explicit contribution, B is the implicit contribution coefficient, and T is the instantaneous temperature of the fluid.
Solid Implicit Force	The implicit portion of the total momentum contribution from particle-fluid interaction forces, which is in turn given by: $A+Bv$ where: A is the explicit contribution, B is the implicit contribution coefficient, and v is the fluid velocity. See also I cannot find the directional properties for Solid Implicit Force (p. 1379).
Solid Implicit Heat Transfer Rate	When **Thermal Model** is enabled (see also Enable Thermal Modeling (p. 400)) and the Fluent case has a thermal solution, this is the implicit heat transfer rate source contribution of the particles into the fluid, where the total heat source is given by: $A+BT$ where: A is the explicit contribution, B is the implicit contribution coefficient, and T is the instantaneous temperature of the fluid.
Solid Volume Fraction	The ratio between the summation of the particle volumes within a CFD cell and the volume of the cell.
Static Pressure	The static pressure of the fluid.

(See also About Using the 2-Way Fluent Method (p. 795)).

EULERIAN STATISTICS PROPERTIES

Figure 5.16: Eulerian Statistics (applied to a Cube), Properties options in the Data Editors panel

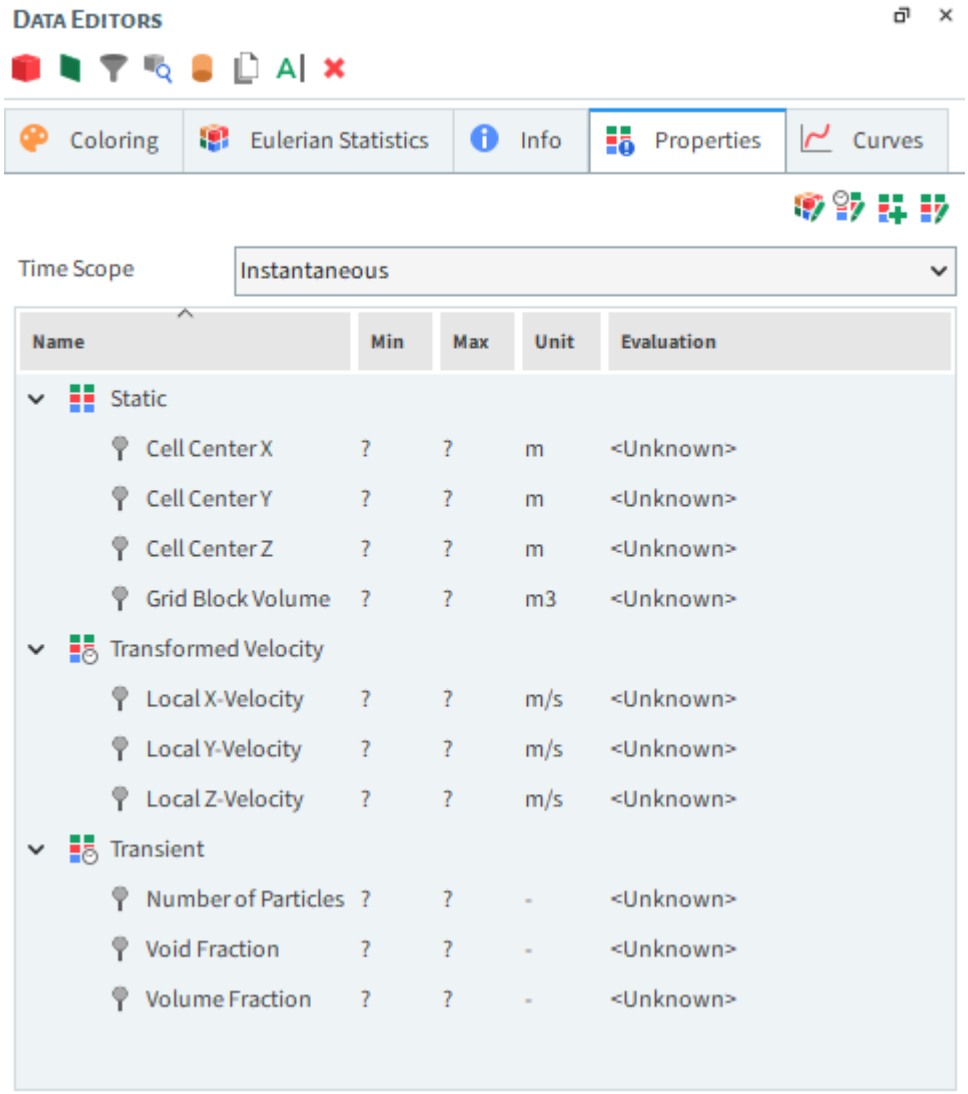


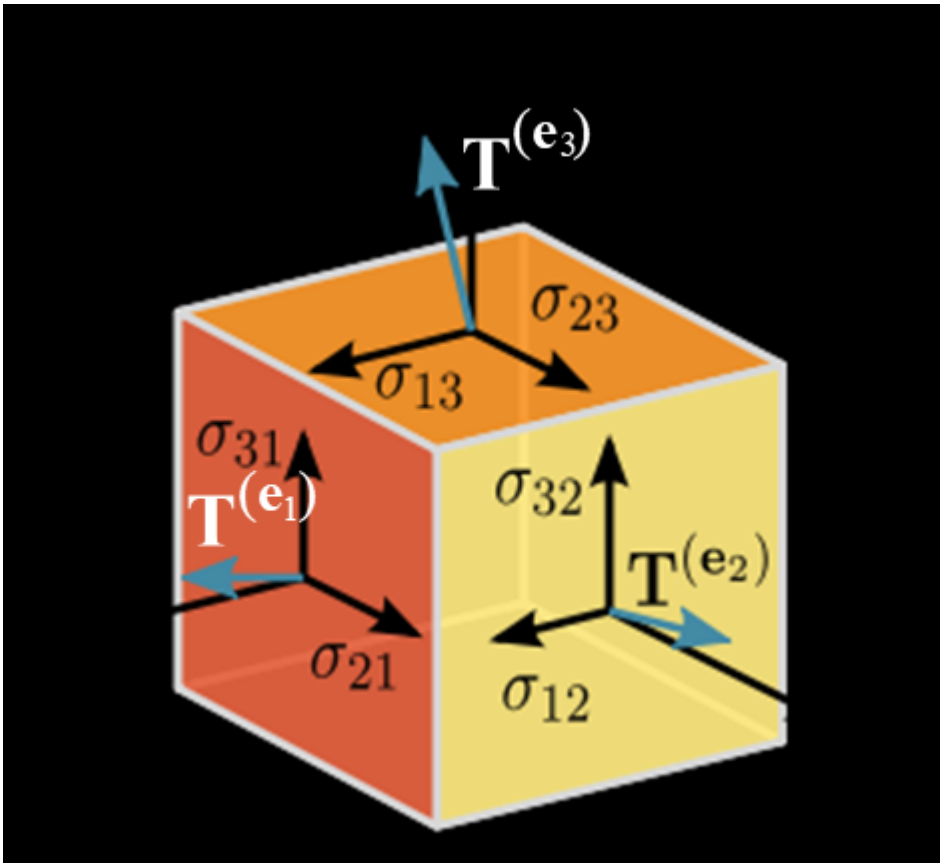
Table 11: Properties available by default for Eulerian Statistics

Property Name	Description
Cell Center X	Eulerian bin coordinates in the X direction. Useful for analyzing properties that can vary spatially, you can create a Cross Plot of a certain property over the bin coordinates in the X direction.
Cell Center Y	Eulerian bin coordinates in the Y direction. Useful for analyzing properties that can vary spatially, you can create a Cross Plot of a certain property over the bin coordinates in the Y direction.
Cell Center Z	Eulerian bin coordinates in the Z direction. Useful for analyzing properties that can vary spatially, you can create a Cross Plot of a certain property over the bin coordinates in the Z direction.
Grid Block Volume	The total volume of each individual block the Cube or Cylinder User Process is divided into (see also About Eulerian Statistics (p. 1226)).

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

Property Name	Description
Stress Component <Various>	**Important:** To obtain this property, you must first have enabled the **Collect Contacts Data** checkbox on the **Contacts** entity prior to processing your simulation (see also About Contacts (p. 746)). The stress tensor for each Eulerian bin is computed as described in the **Stress Tensor** definition (below). Each one of the six independent components represents the average stress tensor resulting from particles' contact with other particles or boundaries. As this is a second order symmetric tensor, three of these values represent normal stresses and three represent shear stresses. Stress components are given in a coordinate system based upon the User Process shape from which the Eulerian Statistics User Process is created. Specifically: • For Cubes, these components are given in a Cartesian coordinate system oriented according to the cube. $S_{xyz}=\begin{bmatrix}\sigma_{xx} & \sigma_{xy} & \sigma_{xz}\\ \sigma_{yx} & \sigma_{yy} & \sigma_{yz}\\ \sigma_{zx} & \sigma_{zy} & \sigma_{zz}\end{bmatrix}$ • For Cylinders, the stress tensor is given using a cylindrical coordinate system that is oriented according to the cylinder. $S\equiv\begin{bmatrix}S_{rr} & S_{r\theta} & S_{rz}\\ S_{\theta r} & S_{\theta\theta} & S_{\theta z}\\ S_{zr} & S_{z\theta} & S_{zz}\end{bmatrix}$ For both Cubes and Cylinders, this follows the conventional index notation (as shown below), whereby stress components have two different subscript indices that vary among X, Y, and Z for Cubes, and R, θ (theta), and Z for Cylinders.

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

Property Name	Description
	Figure 5.17: Conventional index notation for stress tensors within a Cube-shaped selection  • The first index i indicates that the stress acts on a plane normal to the xi-axis • The second index j denotes the direction in which the stress acts A stress component is positive if it acts in the positive direction of the coordinate axes, and if the plane where it acts has an outward normal vector pointing in the positive coordinate direction. This gives the nine components, but since the stress matrix is symmetric, i.e. $\sigma_{ij} = \sigma_{ji}$, only six of these nine components are independent and therefore listed explicitly in Rocky. As an example, in the case of an Eulerian Statistics User Process being applied to a Cube User Process, the stress component YZ indicates the stress on the face whose normal is in the Y direction and acting in the Z direction.
Transformed Velocity	The transformed velocity of each Eulerian bin is computed as: $\mathrm{v}_i = \frac{1}{N_i} \sum_{p=1}^{N_i} \mathrm{v}_p$ where:

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

Property Name	Description
	v_i is the velocity of the bin, i, v_p is the velocity of the particle, p, and N_i is the number of particles in the bin, i.
Local X-Velocity	For Cube User Processes, this is the bin velocity in the local X direction (red directional handle) relative to the Cube, computed based upon the **Transformed Velocity** description (above).
Local Y-Velocity	For Cube User Processes, this is the bin velocity in the local Y direction (green directional handle) relative to the Cube, computed based upon the **Transformed Velocity** description (above).
Local Z-Velocity	For Cube User Processes, this is the bin velocity in the local Z direction (blue directional handle) relative to the Cube, computed based upon the **Transformed Velocity** description (above).
Local R-Velocity	For Cylinder User Processes, this is the particle speed in the radial direction relative to the Cylinder, computed based upon the **Transformed Velocity** description (above).
Local T-Velocity	For Cylinder User Processes, this is the particle speed in the tangential direction relative to the Cylinder, computed based upon the **Transformed Velocity** description (above).
Local Z-Velocity	For Cylinder User Processes, this is the particle speed in the axial direction relative to the Cylinder, computed based upon the **Transformed Velocity** description (above).
First Stress Invariant	**Important:** To obtain this property, you must first have enabled the **Collect Contacts Data** checkbox on the **Contacts** entity prior to processing your simulation (see also About Contacts (p. 746)). Also known as "trace," this is the is the sum of the diagonal stress components, as given by the following equation: $I_1=\sigma_{11}+\sigma_{22}+\sigma_{33}=\sigma_{kk}=tr(\sigma)$
Number of Particles	The amount of particles in each individual block the Cube or Cylinder User Process is divided into (see also About Eulerian Statistics (p. 1226)).
Second Stress Invariant	**Important:** To obtain this property, you must first have enabled the **Collect Contacts Data** checkbox on the **Contacts** entity prior to processing your simulation (see also About Contacts). The sum of the principal minors as given by the following equation: $I_1=\begin{vmatrix}\sigma_{22} & \sigma_{23}\\ \sigma_{32} & \sigma_{33}\end{vmatrix}+\begin{vmatrix}\sigma_{11} & \sigma_{13}\\ \sigma_{31} & \sigma_{33}\end{vmatrix}+\begin{vmatrix}\sigma_{11} & \sigma_{12}\\ \sigma_{21} & \sigma_{22}\end{vmatrix}$ $=\sigma_{11}\sigma_{22}+\sigma_{22}\sigma_{33}+\sigma_{11}\sigma_{33}-\sigma_{12}^2-\sigma_{23}^2-\sigma_{31}^2$ $=\frac{1}{2}\left(\sigma_{ii}\sigma_{jj}-\sigma_{ij}\sigma_{ji}\right)=\frac{1}{2}\left[tr(\sigma)^2-tr(\sigma^2)\right]$
Stress Tensor	**Important:** To obtain this property, you must first have enabled the **Collect Contacts Data** checkbox on the **Contacts** entity prior to processing your simulation (see also About Contacts (p. 746)).

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

Property Name	Description
	The average stress tensor for the bin is computed as: $\sigma_i = \frac{1}{V_i} \sum_{p=1}^{N_i} \sigma_p V_p$ where: v_i is the volume of the bin, i, v_p is the volume of the particle, p, N_i is the number of particles in the bin, i, and σ_p is the stress tensor of particle, p.
Third Stress Invariant	**Important:** To obtain this property, you must first have enabled the **Collect Contacts Data** checkbox on the **Contacts** entity prior to processing your simulation (see also About Contacts (p. 746)). The determinant of the stress tensor, as given by the following equation: $I_3 = \det(\sigma_{ij}) = \det(\sigma)$ $= \sigma_{11}\sigma_{22}\sigma_{33} + 2\sigma_{12}\sigma_{23}\sigma_{31} - \sigma_{12}^2\sigma_{33} - \sigma_{23}^2\sigma_{11} - \sigma_{31}^2\sigma_{22}$
Transformed Stress Tensor	**Important:** To obtain this property, you must first have enabled the **Collect Contacts Data** checkbox on the **Contacts** entity prior to processing your simulation (see also About Contacts (p. 746)). The stress tensor using the local coordinates of the Cylinder or Cube User Process upon which the Eulerian Statistics User Process was created.
Void Fraction	Also known as porosity, this is the ratio between the volume of the block not occupied by particles divided by the volume of the block itself, according to the following equation: $\varepsilon_i = 1 - \phi_i$ where: ϕ_i is the particle volume fraction of the bin, i. **Tip:** **Void Fraction** = 1 - **Volume Fraction**
Volume Fraction	The ratio between the total particle volume inside the block divided by the volume of the block itself, according to the following equation: $\phi_i = \frac{1}{V_i} \sum_{p=1}^{N_i} \frac{m_p}{\rho_p}$ where:

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

Property Name	Description
	v_i is the volume of the bin, i, N_i is the number of particles in the bin,i, m_p is the mass of the particle, p, and ρ_p is the density of the particle, p. Volume Fraction represents the volume of the block occupied by particles. Since it takes into account the sum of the volume of all particles that have nodes located inside the corresponding block, it can return values above 1 in cases where the ratio between the particle size and the block volume is small. **Tip:** **Void Fraction** = 1 - **Volume Fraction**

What would you like to do?

- Learn more About Customizing Properties or Curves (p. 1012)
- Learn more About Customizing Eulerian Statistics Properties (p. 1017)
- Learn more About Adding and Editing Time Statistics Properties (p. 1023)
- Add a New Custom Property on the Properties or Curves Tab (p. 1037)
- Change or Remove an Existing Custom Property on the Properties or Curves Tab (p. 1036)
- Add a Particle Property to a Eulerian Statistics User Process (p. 1037)

See Also:

- Analyzing a Simulation (p. 931)
- About Curves (p. 989)
- I cannot find some Properties or Curves in this version of Rocky (p. 1369)
- About Eulerian Statistics (p. 1226)

5.1.2. About Curves

Unlike Properties, which record many individual values for each *output time* (p. 1393), the Curves tab displays only the entity properties that produce a single value. This means that a Property becomes a Curve after its data set is reduced to a single value; for example, by choosing to sum all the values or average them out.

How Curves are Calculated

Curves are calculated using Global coordinates and are collected for geometry components, including conveyors and imported geometries, as well as for Particles, the Solver, and individual Motion Frames.

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

And as with Properties, individual User Processes linked to any entities with Curve properties will also display the same properties on their own Curves tabs.

About Displaying Curves

Unlike Properties, you cannot display Curves in a 3D View or a Histogram due to the one-dimensional aspect of the data that is recorded. The ways in which you can display Curves in the Rocky UI are described in the table below.

Table 1: Methods for Displaying Curves data

Display Options	Method	See Also
Through the Curves tab	From the property, choose to **Compute Statistics** and then use the UI to display the individual values you want.	View an Individual Statistic (p. 1098)
In a Time Plot	Create a Time Plot from the property.	About Time Plots (p. 1102)

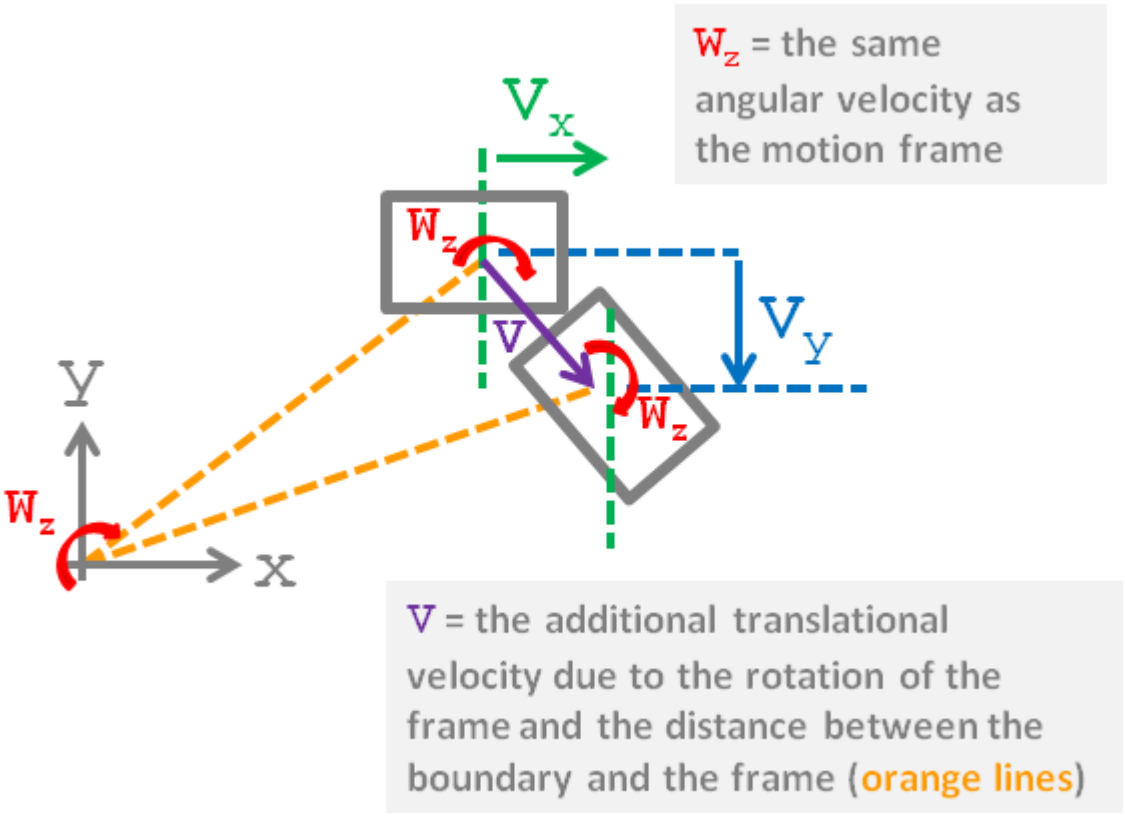
About Curves for Imported Geometries and Motion Frames

In Rocky, both imported Geometries and Motion Frames have Curves properties available, both of which provide translational and angular velocities in different directions. At first glance, it might appear that the velocity values for an imported geometry might be equal to those of the Motion Frame assigned to it, but this is not always the case, as described below and illustrated in Figure 5.18: Diagram showing how Motion Frames assigned to a rectangular boundary affect the resulting velocity Curves for the boundary (p. 991)

- In cases where the Motion Frame assigned to the geometry has only translation motions, the boundary velocities will be equal to that of the Motion Frame.
- In cases where the Motion Frame assigned to the geometry has only rotation motions, the boundary will have the same angular velocity as the Motion Frame plus a separate translational velocity as given by the angular velocity of the frame and the distance between the boundary and the frame (see Figure 1).
- In cases where the Motion Frame assigned to the geometry has both translation and rotation motions, the boundary will have the same angular velocity and a combination of the frame's translational velocity and the additional translational velocity applied due to the frame's rotation and the distance between the boundary and the frame.

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

Figure 5.18: Diagram showing how Motion Frames assigned to a rectangular boundary affect the resulting velocity Curves for the boundary



(See also About Creating and Applying Motion Frames (p. 533) and I cannot find some Properties or Curves in this version of Rocky (p. 1369)).

Definitions of Rocky Curves

See the images and tables below to learn about the various Curves available in Rocky.

GEOMETRY COMPONENT CURVES

(See also About Collision Statistics for Boundaries (p. 312))

Figure 5.19: Geometry (Feed; Receiving; Imported) Curves options on the Data Editors panel when Boundary Collision Statistics are collected

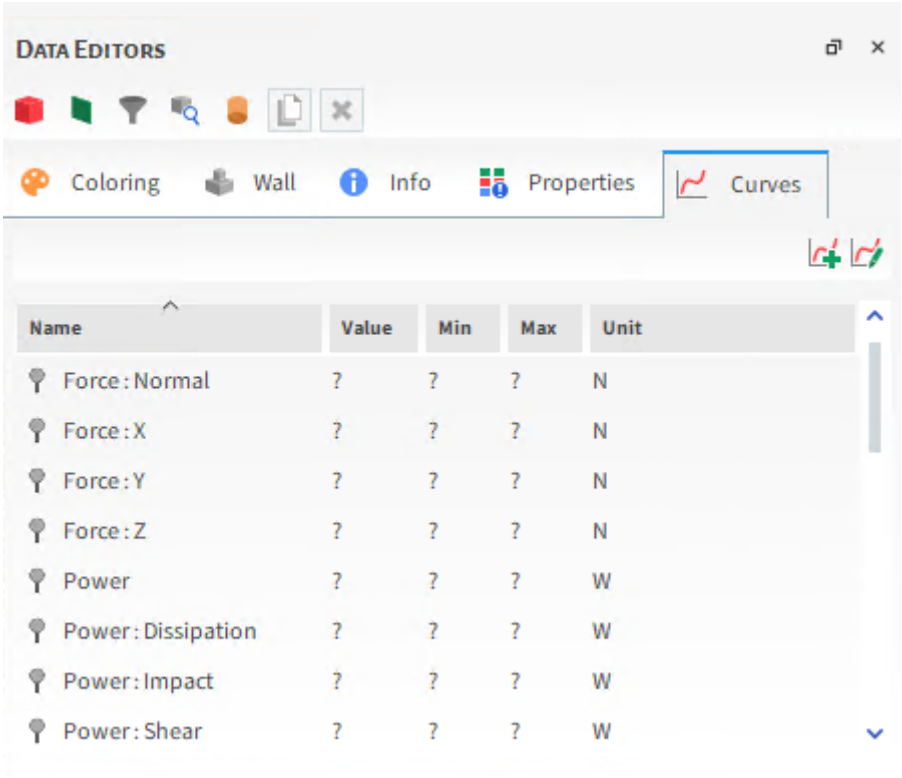
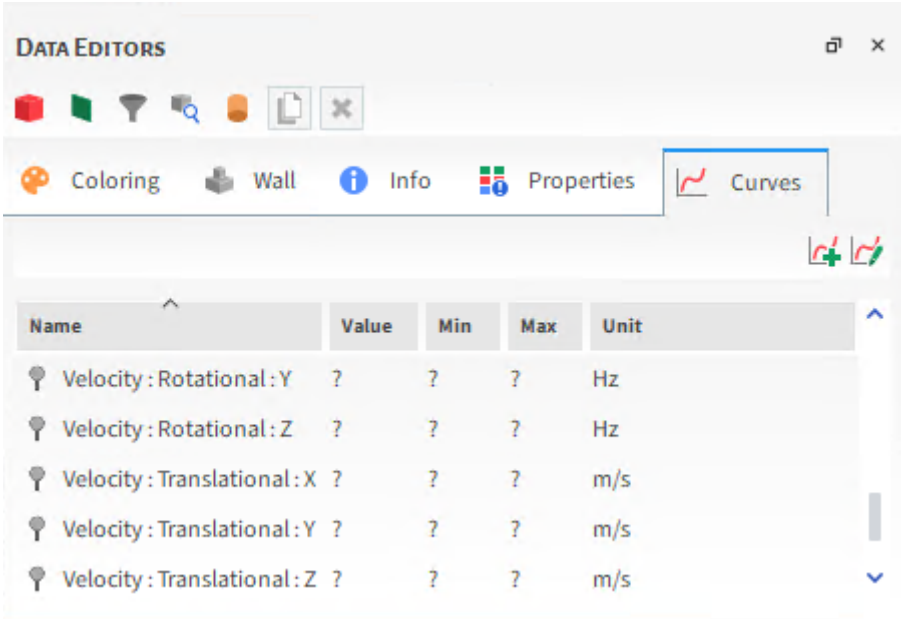


Figure 5.20: Imported Geometry, Curves options on the Data Editors panel when Motion Frames are assigned



(See also About Creating and Applying Motion Frames (p. 533) and I cannot find some Properties or Curves in this version of Rocky (p. 1369).)

Table 2: Curves options for Geometries (Feed Conveyors, Receiving Conveyors, and Imported)

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

Function	Description
All Geometries When Boundary Collision Statistics are collected Tip: See also About Collision Statistics for Boundaries (p. 312).	
Force : Normal	When the **Stresses** option is enabled on the **Boundary Collision Statistics** module (see also About Collision Statistics for Boundaries (p. 312)), this provides the sum of the normal forces acting on all the triangles that make up a geometry surface. This is useful for measuring the degree of upwards belt misalignment.
Force : X	When the **Forces for FEM Analysis** option is enabled on the **Boundary Collision Statistics** module (see also About Collision Statistics for Boundaries (p. 312)), this provides the force applied to the geometry along the X axis. This is useful for measuring the degree of belt misalignment in the X direction.
Force : Y	When the **Forces for FEM Analysis** option is enabled on the **Boundary Collision Statistics** module (see also About Collision Statistics for Boundaries (p. 312)), this provides the force applied to the geometry along the Y axis. This is useful for measuring the degree of belt misalignment in the Y direction.
Force : Z	When the **Forces for FEM Analysis** option is enabled on the **Boundary Collision Statistics** module (see also About Collision Statistics for Boundaries (p. 312)), this provides the force applied to the geometry along the Z axis. This is useful for measuring the degree of belt misalignment in the Z direction.
Power	When the **Intensities** option is enabled on the **Boundary Collision Statistics** module (see also About Collision Statistics for Boundaries (p. 312)), this provides the power required to move the geometry at the specified speed given the particle load applied to the surface. This is useful for determining the power applied by the geometry.
Power : Dissipation	When the **Intensities** option is enabled on the **Boundary Collision Statistics** module (see also About Collision Statistics for Boundaries (p. 312)), this provides the power dissipated in the collisions against a geometry surface by the contact forces. This is useful for determining the amount of mechanical energy that is transformed irreversibly into other forms of energy during a collision.
Power : Impact	When the **Intensities** option is enabled on the **Boundary Collision Statistics** module (see also About Collision Statistics for Boundaries (p. 312)), this provides the power transferred during the loading phase of all collisions against the geometry surface. It is related to the normal component of the contact force, and it is useful for determining how particle impact affects the surface
Power : Shear	When the **Intensities** option is enabled on the **Boundary Collision Statistics** module (see also About Collision Statistics for

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

Function	Description
	Boundaries (p. 312)), this provides the power transferred during the unloading phase of all collisions against the geometry surface. This is useful for predicting the abrasive wear of surface boundaries.
Imported Geometries Only When Motion Frames are assigned Tip:See additional information in the *About Curves for Imported Geometries and Motion Frames* section above	
Velocity : Rotational : X	When the geometry has a Motion Frame assigned to it (see also About Creating and Applying Motion Frames to Custom Geometries (p. 533)), this is the rotational (angular) velocity of the assigned Motion Frame as defined in the X direction.
Velocity : Rotational : Y	When the geometry has a Motion Frame assigned to it (see also About Creating and Applying Motion Frames to Custom Geometries (p. 533)), this is the rotational (angular) velocity of the assigned Motion Frame as defined in the Y direction.
Velocity : Rotational : Z	When the geometry has a Motion Frame assigned to it (see also About Creating and Applying Motion Frames to Custom Geometries (p. 533)), this is the rotational (angular) velocity of the assigned Motion Frame as defined in the Z direction.
Velocity : Translational : X	When the geometry has a Motion Frame assigned to it (see also About Creating and Applying Motion Frames to Custom Geometries (p. 533)), this is the translational velocity of the assigned Motion Frame as defined in the X direction.
Velocity : Translational : Y	When the geometry has a Motion Frame assigned to it (see also About Creating and Applying Motion Frames to Custom Geometries (p. 533)), this is the translational velocity of the assigned Motion Frame as defined in the Y direction.
Velocity : Translational : Z	When the geometry has a Motion Frame assigned to it (see also About Creating and Applying Motion Frames to Custom Geometries (p. 533)), this is the translational velocity of the assigned Motion Frame as defined in the Z direction.

STANDARD CURVES FOR PARTICLES

Notes:

- Curves related to particle breakage are applicable only for simulations with *Instantaneous Breakage* (p. 1392).
- For most Curves, a point in an *output time* (p. 1393) represents what happened in the interval between that output time and the previous one.

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

(See also About Adding and Editing Particle Sets (p. 602).)

Figure 5.21: Particles, Curves options on the Data Editors panel

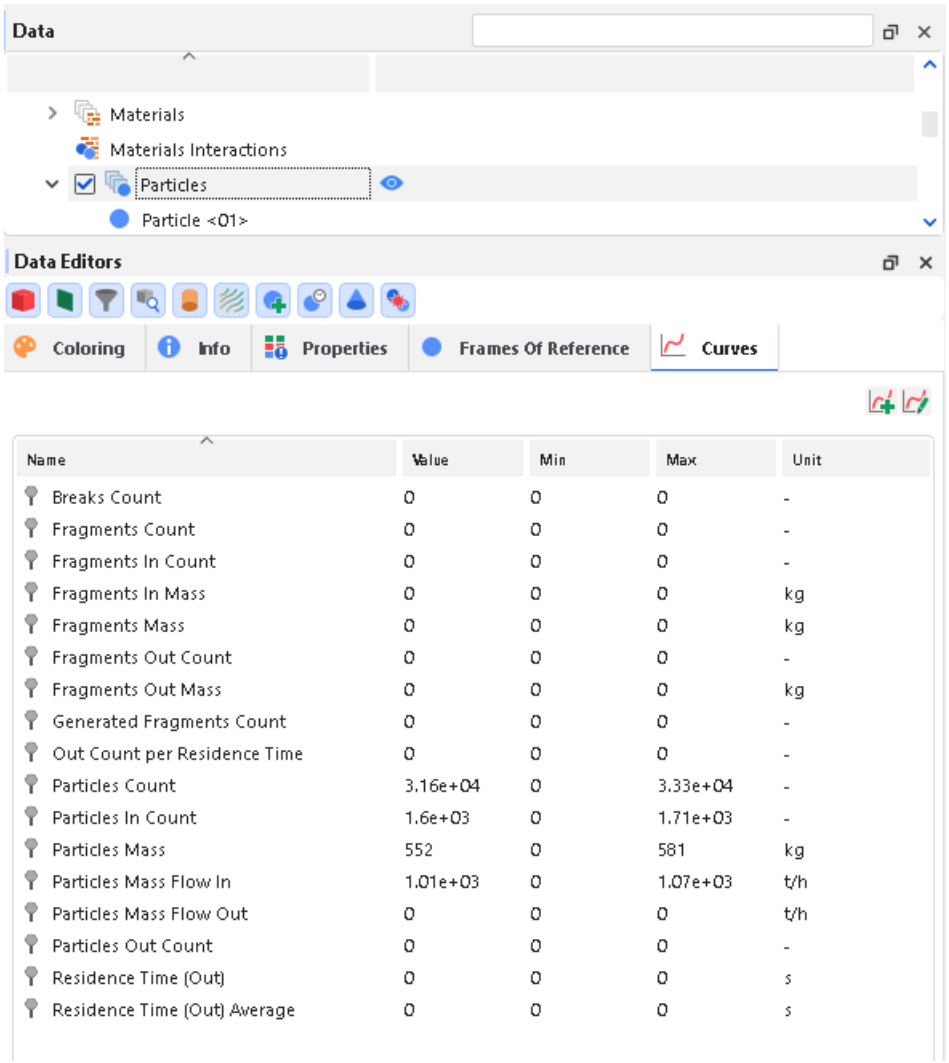


Table 3: Standard Curves options for Particles

Function	Description
Breaks Count	When *Instantaneous Breakage* (p. 1392) parameters are set, this provides the total number of individual whole particles and *fragments* that broke during an interval between two consecutive output times.
Fragments Count	When *Instantaneous Breakage* (p. 1392) parameters are set, this provides the total number of fragments that are currently in the selection.
Fragments In Count	When *Instantaneous Breakage* (p. 1392) parameters are set, this provides the total number of particle fragments that entered the selection during an interval between two consecutive output times.
Fragments In Mass	When *Instantaneous Breakage* (p. 1392) parameters are set, this provides the total mass of particle fragments that have entered the selection during an interval between two consecutive output times.

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

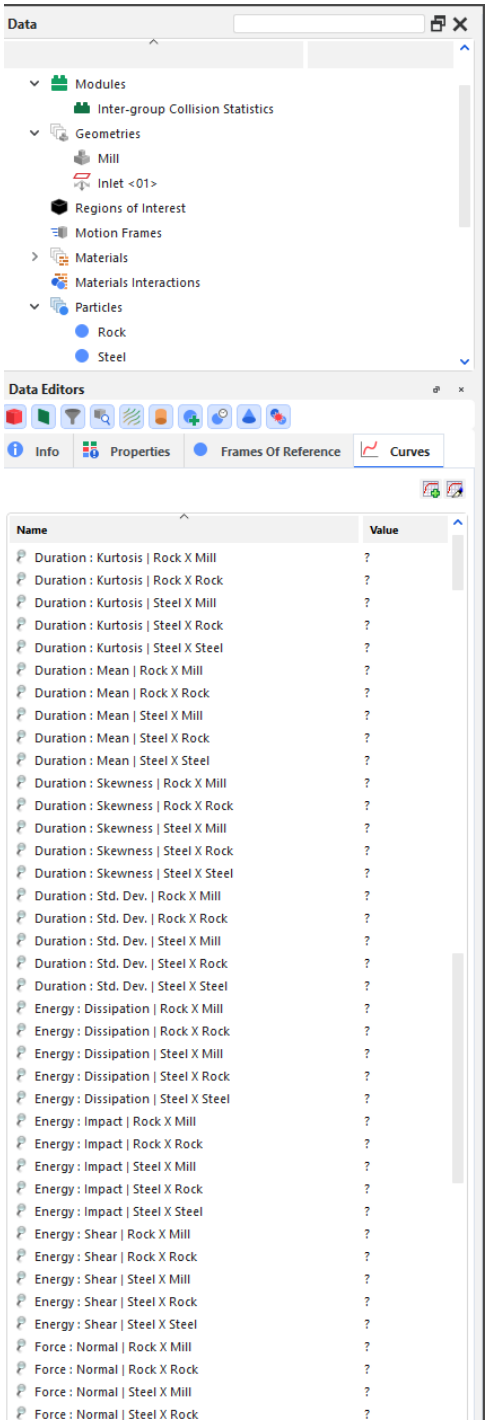
Function	Description
Fragments Mass	When *Instantaneous Breakage* (p. 1392) parameters are set, this provides the total mass of the particle fragments that are currently within the selection.
Fragments Out Count	When *Instantaneous Breakage* (p. 1392) parameters are set, this provides the total number of particle fragments that have exited the selection during an interval between two consecutive output times.
Fragments Out Mass	When *Instantaneous Breakage* (p. 1392) parameters are set, this provides the total mass of particle fragments that have exited the selection during an interval between two consecutive output times.
Generated Fragments Count	When *Instantaneous Breakage* (p. 1392) parameters are set, this provides the total number of new particle fragments that were generated inside the selection during an interval between two consecutive output times.
Out Count per Residence Time	For each point in the curve, the vertical coordinate indicates the total number of whole particles and fragments inside the selection that have spent the time represented by the horizontal coordinate.
Particles Count	The total number of whole particles and particle fragments that are currently in the selection.
Particles In Count	The total number of whole particles and particle fragments that have entered the selection during an interval between two consecutive output times.
Particles Mass Flow In	The mass flow of particles and fragments entering the selection in a given simulation output.
Particles Mass	The total mass of whole particles and fragments that are currently in the selection.
Particles Out Count	The total number of whole particles and fragments that have exited the selection during an interval between two consecutive output times.
Particles Mass Flow Out	The mass flow of particles and fragments exiting the selection in a given simulation output.
Residence Time (Out)	For any whole particles or fragments that have exited the selection during an interval between two consecutive output times, this is the average amount of time that they were inside the selection boundaries before exiting.
Residence Time (Out) Average	For all whole particles and fragments that have exited the selection prior to the last fully calculated output time, this is the average (but cumulative) amount of time that they were inside the simulation boundaries before exiting.

INTER-GROUP COLLISION STATISTICS CURVES FOR PARTICLES

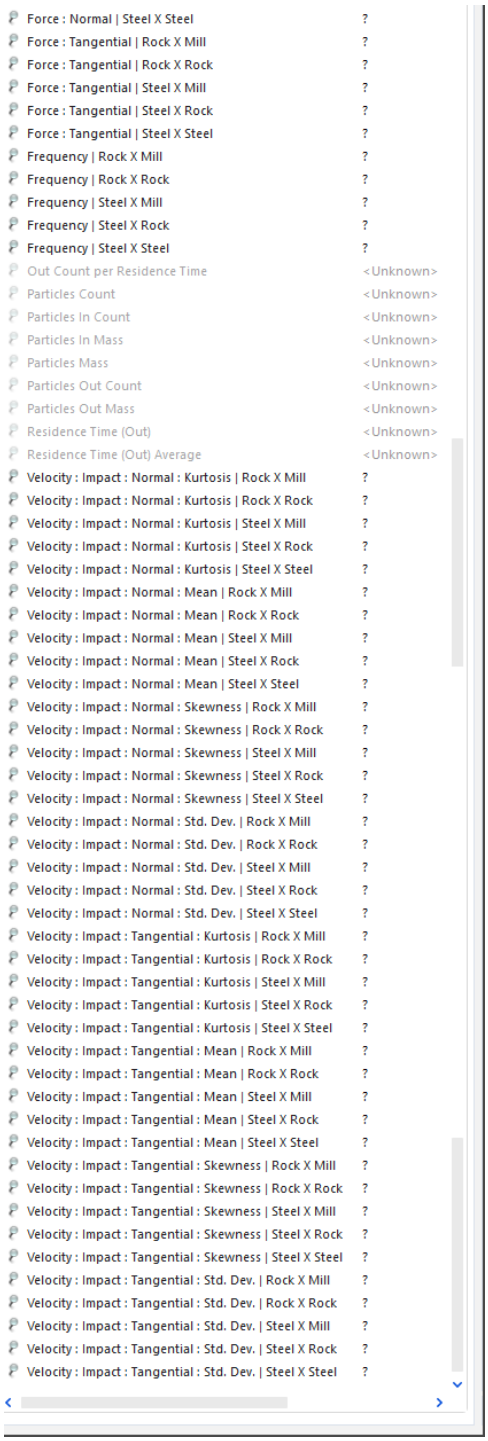
(See also About Inter-group Collision Statistics (p. 319).)

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

Figure 5.22: Particles, Curves options on the Data Editors panel when Inter-group Collision Statistics is enabled



Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.



Note:

For all Inter-group Collisions Statistics curves, the effects of upscaling will be ignored for any *meshed* (p. 1392) particles with **Meshed Particles Upscaling** enabled. (See also About Meshed Particles Upscaling (p. 1033)).

Table 4: Additional Curves options for Particles when Inter-group Collision Statistics is enabled

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

Function	Description
Duration : Kurtosis	When the **Duration** option is enabled on the **Inter-group Collision Statistics** module (see also About Inter-group Collision Statistics (p. 319)), this provides the kurtosis value of the duration of all collisions recorded for the indicated pair (particle-particle or particle-geometry) that occurred during an interval between two consecutive *output time* (p. 1393). The kurtosis is a measure of how tall and sharp the central peak of a statistical distribution is. See also the "Collision statistics" section of the DEM Technical Manual (from the Rocky **Help** menu, point to **Manuals**, and then click **DEM Technical Manual**).
Duration : Mean	When the **Duration** option is enabled on the **Inter-group Collision Statistics** module (see also About Inter-group Collision Statistics (p. 319)), this provides the mean value of the duration of all collisions recorded for the indicated pair (particle-particle or particle-geometry) that occurred during an interval between two consecutive output times. See also the "Collision statistics" section of the DEM Technical Manual (from the Rocky **Help** menu, point to **Manuals**, and then click **DEM Technical Manual**).
Duration : Skewness	When the **Duration** option is enabled on the **Inter-group Collision Statistics** module (see also About Inter-group Collision Statistics (p. 319)), this provides the skewness value of the duration of all collisions recorded for the indicated pair (particle-particle or particle-geometry) that occurred during an interval between two consecutive output times. The skewness is a measure of the asymmetry of a statistical distribution about its mean. See also the "Collision statistics" section of the DEM Technical Manual (from the Rocky **Help** menu, point to **Manuals**, and then click **DEM Technical Manual**).
Duration : Std. Dev.	When the **Duration** option is enabled on the **Inter-group Collision Statistics** module (see also About Inter-group Collision Statistics (p. 319)), this provides the standard deviation value of the duration of all collisions recorded for the indicated pair (particle-particle or particle-geometry) that occurred during an interval between two consecutive output times. See also the "Collision statistics" section of the DEM Technical Manual (from the Rocky **Help** menu, point to **Manuals**, and then click **DEM Technical Manual**).
Energy : Dissipation	When the **Energy Dissipation** option is enabled on the **Inter-group Collision Statistics** module (see also About Inter-group Collision Statistics (p. 319)), this provides the total energy dissipated for all collisions recorded for the indicated pair (particle-particle or particle-geometry) that occurred during an interval between two consecutive output times. See also the "Collision statistics" section of the DEM Technical Manual (from the Rocky **Help** menu, point to **Manuals**, and then click **DEM Technical Manual**).
Energy : Impact	When the **Impact Energy** option is enabled on the **Inter-group Collision Statistics** module (see also About Inter-group Collision Statistics (p. 319)), this provides the total impact energy for all collisions recorded for the indicated pair (particle-particle or particle-geometry) that occurred during an interval between two consecutive output times.

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

Function	Description
	See also the "Collision statistics" section of the DEM Technical Manual (from the Rocky **Help** menu, point to **Manuals**, and then click **DEM Technical Manual**).
Energy : Shear	When the **Shear Energy** option is enabled on the **Inter-group Collision Statistics** module (see also About Inter-group Collision Statistics (p. 319)), this provides the total shear energy for all collisions recorded for the indicated pair (particle-particle or particle-geometry) that occurred during an interval between two consecutive output times. See also the "Collision statistics" section of the DEM Technical Manual (from the Rocky **Help** menu, point to **Manuals**, and then click **DEM Technical Manual**).
Force : Normal	When the **Forces** option is enabled on the **Inter-group Collision Statistics** module (see also About Inter-group Collision Statistics (p. 319)), this provides the total force in the normal direction for all collisions recorded for the indicated pair (particle-particle or particle-geometry) that occurred during an interval between two consecutive output times. See also the "Collision statistics" section of the DEM Technical Manual (from the Rocky **Help** menu, point to **Manuals**, and then click **DEM Technical Manual**).
Force : Tangential	When the **Forces** option is enabled on the **Inter-group Collision Statistics** module (see also About Inter-group Collision Statistics (p. 319)), this provides the total force in the tangential direction for all collisions recorded for the indicated pair (particle-particle or particle-geometry) that occurred during an interval between two consecutive output times. See also the "Collision statistics" section of the DEM Technical Manual (from the Rocky **Help** menu, point to **Manuals**, and then click **DEM Technical Manual**).
Frequency	When the **Frequency** option is enabled on the **Inter-group Collision Statistics** module (see also About Inter-group Collision Statistics (p. 319)), this provides the total collision frequency for all collisions recorded for the indicated pair (particle-particle or particle-geometry) that occurred during an interval between two consecutive output times. See also the "Collision statistics" section of the DEM Technical Manual (from the Rocky **Help** menu, point to **Manuals**, and then click **DEM Technical Manual**).
Velocity : Impact : Normal : Kurtosis	When the **Normal Impact Velocity** option is enabled on the **Inter-group Collision Statistics** module (see also About Inter-group Collision Statistics (p. 319)), this provides the kurtosis value of the impact relative velocity in normal direction for all collisions recorded for the indicated pair (particle-particle or particle-geometry) that occurred during an interval between two consecutive output times. The kurtosis is a measure of how tall and sharp the central peak of a statistical distribution is. See also the "Collision statistics" section of the DEM Technical Manual (from the Rocky **Help** menu, point to **Manuals**, and then click **DEM Technical Manual**).
Velocity : Impact : Normal : Mean	When the **Normal Impact Velocity** option is enabled on the **Inter-group Collision Statistics** module (see also About Inter-group Collision Statistics (p. 319)), this provides the mean value of the impact relative velocity in normal direction for all collisions recorded for the indicated pair

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

Function	Description
	(particle-particle or particle-geometry) that occurred during an interval between two consecutive output times. See also the "Collision statistics" section of the DEM Technical Manual (from the Rocky **Help** menu, point to **Manuals**, and then click **DEM Technical Manual**).
Velocity : Impact : Normal : Skewness	When the **Normal Impact Velocity** option is enabled on the **Inter-group Collision Statistics** module (see also About Inter-group Collision Statistics (p. 319)), this provides the skewness value of the impact relative velocity in normal direction for all collisions recorded for the indicated pair (particle-particle or particle-geometry) that occurred during an interval between two consecutive output times. The skewness is a measure of the asymmetry of a statistical distribution about its mean. See also the "Collision statistics" section of the DEM Technical Manual (from the Rocky **Help** menu, point to **Manuals**, and then click **DEM Technical Manual**).
Velocity : Impact : Normal : Std. Dev.	When the **Normal Impact Velocity** option is enabled on the **Inter-group Collision Statistics** module (see also About Inter-group Collision Statistics (p. 319)), this provides the standard deviation value of the impact relative velocity in normal direction for all collisions recorded for the indicated pair (particle-particle or particle-geometry) that occurred during an interval between two consecutive output times. See also the "Collision statistics" section of the DEM Technical Manual (from the Rocky **Help** menu, point to **Manuals**, and then click **DEM Technical Manual**).
Velocity : Impact : Tangential : Kurtosis	When the **Tangential Impact Velocity** option is enabled on the **Inter-group Collision Statistics** module (see also About Inter-group Collision Statistics (p. 319)), this provides the kurtosis value of the impact relative velocity in tangential direction for all collisions recorded for the indicated pair (particle-particle or particle-geometry) that occurred during an interval between two consecutive output times. The kurtosis is a measure of how tall and sharp the central peak of a statistical distribution is. See also the "Collision statistics" section of the DEM Technical Manual (from the Rocky **Help** menu, point to **Manuals**, and then click **DEM Technical Manual**).
Velocity : Impact : Tangential : Mean	When the **Tangential Impact Velocity** option is enabled on the **Inter-group Collision Statistics** module (see also About Inter-group Collision Statistics (p. 319)), this provides the mean value of the impact relative velocity in tangential direction for all collisions recorded for the indicated pair (particle-particle or particle-geometry) that occurred during an interval between two consecutive output times. See also the "Collision statistics" section of the DEM Technical Manual (from the Rocky **Help** menu, point to **Manuals**, and then click **DEM Technical Manual**).
Velocity : Impact : Tangential : Skewness	When the **Tangential Impact Velocity** option is enabled on the **Inter-group Collision Statistics** module (see also About Inter-group Collision Statistics (p. 319)), this provides the skewness value of the impact relative velocity in tangential direction for all collisions recorded for the indicated pair (particle-particle or particle-geometry) that occurred during an interval between

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

Function	Description
	two consecutive output times. The skewness is a measure of the asymmetry of a statistical distribution about its mean. See also the "Collision statistics" section of the DEM Technical Manual (from the Rocky **Help** menu, point to **Manuals**, and then click **DEM Technical Manual**).
Velocity : Impact : Tangential : Std. Dev.	When the **Tangential Impact Velocity** option is enabled on the **Inter-group Collision Statistics** module (see also About Inter-group Collision Statistics (p. 319)), this provides the standard deviation value of the impact relative velocity in tangential direction for all collisions recorded for the indicated pair (particle-particle or particle-geometry) that occurred during an interval between two consecutive output times. See also the "Collision statistics" section of the DEM Technical Manual (from the Rocky **Help** menu, point to **Manuals**, and then click **DEM Technical Manual**).

PARTICLE INSTANTANEOUS ENERGIES CURVES FOR PARTICLES

(See also About Particle Instantaneous Energies (p. 341).)

Figure 5.23: Particles, Curves options when Particle Instantaneous Energies is collected

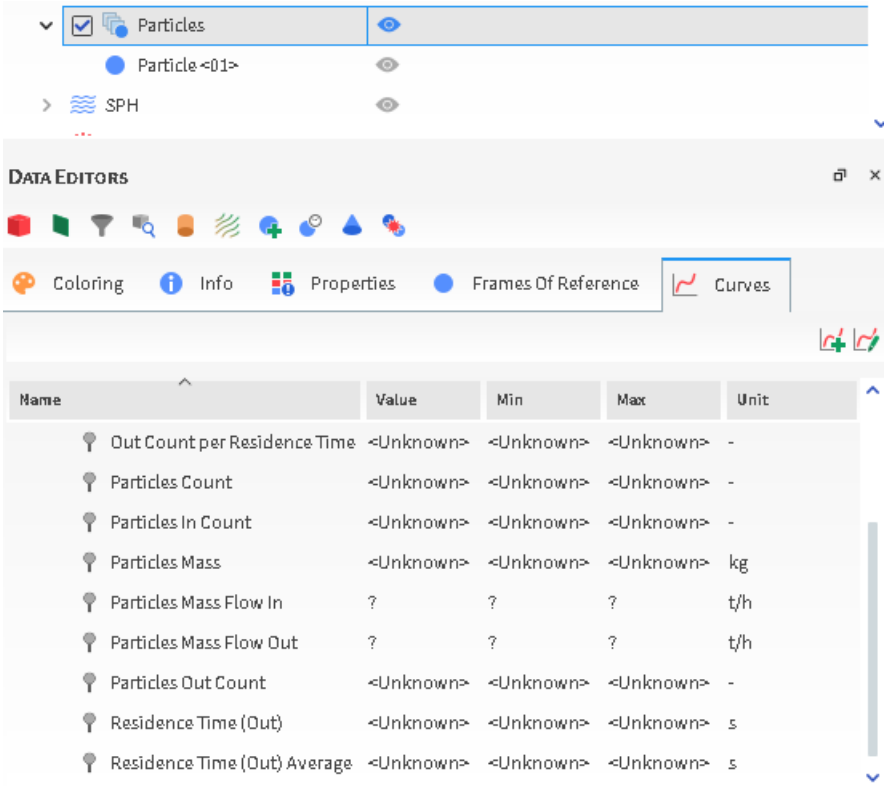


Table 5: Particle Curves options for Particle Instantaneous Energies

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

Function	Description
Energy Delta	When the **Particle Instantaneous Energies** Module is enabled (see also About Particle Instantaneous Energies (p. 341)), this provides a variation (delta) in total energy of the particles recorded during an interval between two consecutive *output time* (p. 1393). Notes: • The value for the first output time is always undefined. • The value for two consecutive output times with different numbers of particles is undefined, because the way that the curve is computed is only meaningful in systems with a constant number of particles. **Tip:** This Curve can be especially useful when computed in User Processes created on the Particles as it will represent the energy delta in only that subset of particles.

ENERGY SPECTRA CURVES FOR PARTICLES

(See also About Particles Energy Spectra (p. 358) and About Contacts Energy Spectra (p. 350))

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

Figure 5.24: Particles, Curves options when Energy Spectra is collected

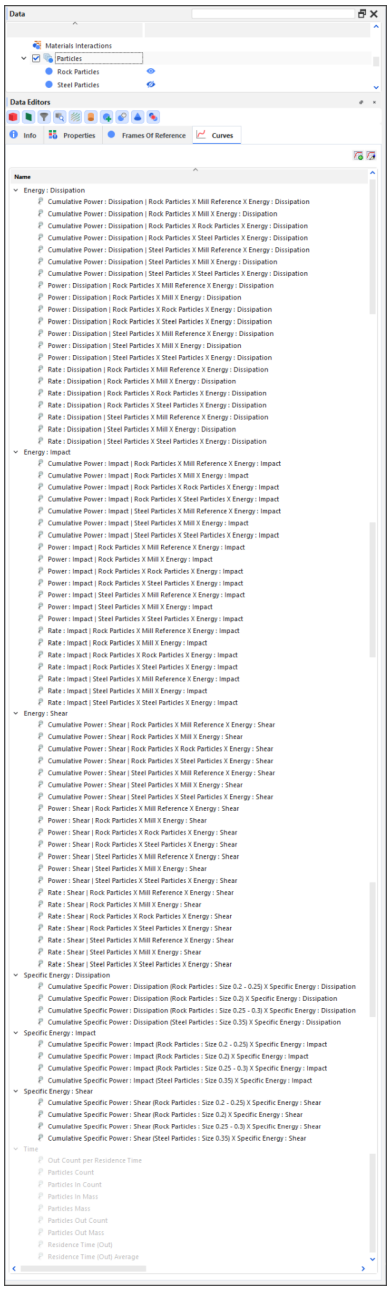


Table 6: Particle Curves options for Energy Spectra

Function	Description
Energy Types	
Energy : Dissipation (Specific Energy : Dissipation)	When the **Particles Energy Spectra** module and/or the **Contacts Energy Spectra** module is enabled (see also About Particles Energy Spectra (p. 358) and About Contacts Energy Spectra (p. 350)), and the **Dissipation Energy** checkbox on that module is enabled, this provides the dissipated energy component of the specified curve.

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

Function	Description
Energy : Impact (Specific Energy : Impact)	When the **Particles Energy Spectra** module and/or the **Contacts Energy Spectra** module is enabled (see also About Particles Energy Spectra (p. 358) and About Contacts Energy Spectra (p. 350)), and the **Impact Energy** checkbox on that module is enabled, this provides the impact energy component of the specified curve.
Energy : Shear (Specific Energy : Shear)	When the **Particles Energy Spectra** module and/or the **Contacts Energy Spectra** module is enabled (see also About Particles Energy Spectra (p. 358) and About Contacts Energy Spectra (p. 350)), and the **Shear Energy** checkbox on that module is enabled, this provides the shear energy component of the specified curve.
Energy Curves	
Cumulative Power	When the **Contacts Energy Spectra** module is enabled (see also About Contacts Energy Spectra (p. 350)), this provides for each energy type collected (Dissipation, Impact, and/or Shear) and contact pair (Particle-Particle or Particle-Geometry) indicated, the sum of all energy coming from a collision with energy greater than a specified energy level divided by the amount of time existing between the module's Start Time and the current output time.
Power	When the **Contacts Energy Spectra** module is enabled (see also About Contacts Energy Spectra (p. 350)), this provides for each energy type collected (Dissipation, Impact, and/or Shear) and contact pair (Particle-Particle or Particle-Geometry) indicated, the time average of energy resulting from all collisions that occur between the module's Start Time and the current output time.
Rate	When the **Contacts Energy Spectra** module is enabled (see also About Contacts Energy Spectra (p. 350)), this provides for each energy type collected (Dissipation, Impact, and/or Shear) and contact pair (Particle-Particle or Particle-Geometry) indicated, the mean frequency of collision throughout the module's Start Time and the current output time.
Cumulative Specific Power	When the **Particles Energy Spectra** module is enabled (see also About Particles Energy Spectra (p. 358)), this provides for each Specific Energy type collected (Dissipation, Impact, and/or Shear) and Particle set - Particle Size group indicated, the sum of all specific energy coming from a collision with energy greater than a specified energy level divided by the amount of time existing between the module's Start Time and the current output time.

SOLVER CURVES

(See also About Solver Parameters (p. 464).)

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

Figure 5.25: Solver, Curves options on the Data Editors panel

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

Table 7: Curves options for the Solver entity

Function	Description
Contacts Adhesive	The total number of particle-to-particle and particle-to-boundary contacts resulting from adhesion that have been made. (For more information about Adhesive Forces, see About Physics Parameters (p. 417)).
Contacts Adhesive Geometry	The total number of particle-to-boundary contacts resulting from adhesion that have been made. (For more information about Adhesive Forces, see About Physics Parameters (p. 417)).
Contacts Adhesive Particle	The total number of particle-to-particle contacts resulting from adhesion that have been made. (For more information about Adhesive Forces, see About Physics Parameters (p. 417)).
Contacts Frictional	The total number of particle-to-particle and particle-to-boundary contacts in which the entities are physically touching each other.
Contacts Frictional Geometry	The total number of particle-to-boundary contacts in which the entities are physically touching each other.
Contacts Frictional Particle	The total number of particle-to-particle contacts in which the entities are physically touching each other.
CPU Memory Free	The amount of memory available on the CPU.
CPU Memory Total	The total amount of memory—both used and unused—the CPU has.
CPU Memory Usage	The amount of memory that is being used on the CPU.
Enabled Coupled SPH Elements	The total amount of SPH elements figured inside the particles for the purpose of interacting with external SPH elements in an SPH-DEM interaction simulation.
Enabled SPH Elements	The total number of SPH Elements that are currently located within the simulation boundaries.
Execution Total Time	The amount of time (duration) that has passed since the simulation started processing again from a stopped state.
GPU Count	The total number of GPU cards that are processing the simulation.
GPU Memory Free	The amount of memory available on the GPU(s) that is processing the simulation.
GPU Memory Total	The total amount of memory—both used and unused—that the GPU(s) processing the simulation has.
GPU Memory Usage	The amount of memory being used on the GPU(s) processing the simulation.
Halo Particles: Ratio	The ratio between the number of halo particles and the total number of particles in the domain.
Halo SPH Elements	In multi-GPU runs, Halo SPH Elements are those that are located in the overlapping devices domain regions and therefore are present simultaneously in two GPU devices. For the Halo SPH Elements, both devices compute the collision forces but only one of the devices accumulates these data in order to perform velocity and displacement calculations.

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

Function	Description
IO Fraction	The fraction of each unit of simulation time allotted to I/O (input/output), which involves writing to and reading from the simulation information on disk. This fraction is a value between 0.0 and 1.0: when closer to 0.0, less wall clock time was spent on I/O.
Neighbor Computation Fraction	The fraction of each unit of simulation time allotted to the counting of neighbor particles, which in turn help identify possible collisions. This is a value between 0.0 and 1.0: when closer to 0.0, less wall clock time was spent searching for neighboring particles.
Neighbor Count	The total amount of neighboring particles detected. This value indicates the number of possible collision events.
Particles Elements	For particles composed of multiple *elements* (p. 1390) (also known as *meshed*), this is the total number of individual elements that will be injected into the simulation during the entire Simulation Duration.
Particle Elements Enabled	For particles composed of multiple *elements* (p. 1390) (also known as *meshed*), this is the total number of individual elements that are currently located within the simulation boundaries.
Particle Elements Released	For particles composed of multiple *elements* (p. 1390) (also known as *meshed*), this is the total number of individual elements that have appeared in the simulation thus far.
Particles Enabled	The total number of whole particles that are currently located within the simulation boundaries. This is useful for determining breakage efficiency. **Note:** Once a whole particle is broken into *fragments* (p. 1393), it is no longer counted towards this value.
Particle Fragments	When *Instantaneous Breakage* (p. 1392) parameters are set, this provides the total number of individual *fragments* (p. 1393) the whole particles have been broken into. This is useful for determining breakage efficiency.
Particles Released	The total amount of all whole particles that have appeared in the simulation.
Particles Total	The total number of whole particles that will be injected into the simulation during the entire **Simulation Duration**. (See also About Solver Parameters (p. 464)). This includes particles that have yet to be released but does not include *fragments* (p. 1393) resulting from *Instantaneous Breakage* (p. 1392).
Simulation ETA	The estimated amount of time (duration) required to finish processing the simulation. This value considers the current simulation configuration.
Simulation Output Index	The *output file* (p. 1393) saved for the *output time* (p. 1393) (See also Refer to the Rocky Title Bar for Simulation Progress Details (p. 119)). This is

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

Function	Description
	useful for determining at what point during the simulation an event of interest occurred.
Simulation Pace	The estimated amount of time (duration) required to advance the simulation one second. This value considers the current simulation configuration.
Halo SPH Elements Ratio	The ratio between the number of halo elements and the total number of SPH Elements in the domain.
Total Elapsed Simulation Time	The total amount of time (duration) that the simulation has been processing since it was originally started. This is calculated by adding up the various Execution Total Times from multiple, separate executions (stops and resumes) of the simulation.

MOTION FRAMES CURVES

(See also About Creating and Applying Motion Frames (p. 533) and I cannot find some Properties or Curves in this version of Rocky (p. 1369)). Tip:See additional information in the *About Curves for Imported Geometries and Motion Frames* section above.

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

Figure 5.26: Motion Frames, Curves options on the Data Editors panel

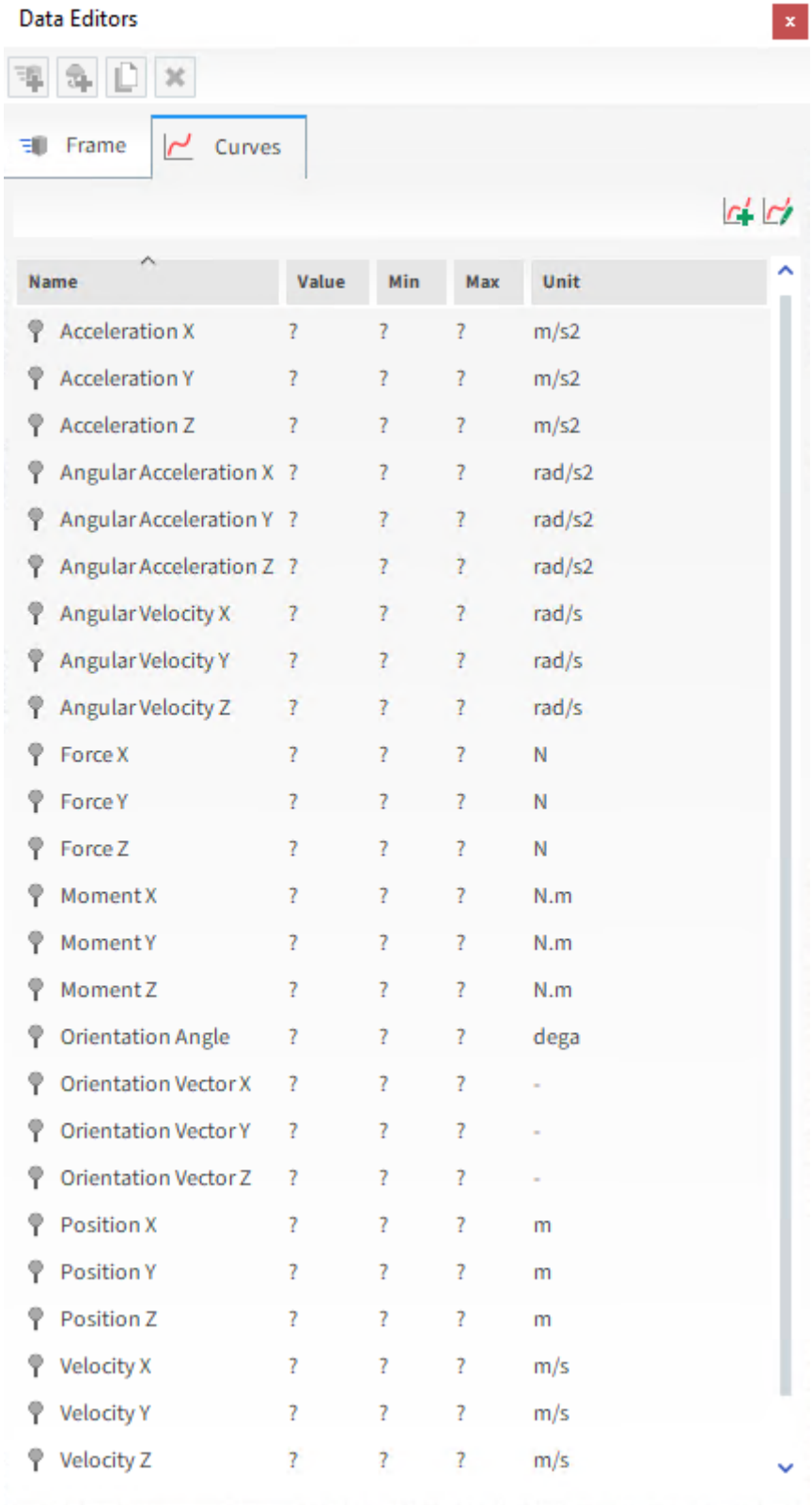


Table 8: Curves options for Motion Frames

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

Function	Description
Acceleration X	The motion frame's acceleration along the X axis.
Acceleration Y	The motion frame's acceleration along the Y axis.
Acceleration Z	The motion frame's acceleration along the Z axis.
Angular Acceleration X	The X-component of the motion frame's angular acceleration vector.
Angular Acceleration Y	The Y-component of the motion frame's angular acceleration vector.
Angular Acceleration Z	The Z-component of the motion frame's angular acceleration vector.
Angular Velocity X	The X-component of the motion frame's angular velocity vector.
Angular Velocity Y	The Y-component of the motion frame's angular velocity vector.
Angular Velocity Z	The Z-component of the motion frame's angular velocity vector.
Force X	The force applied to the motion frame along the X axis.
Force Y	The force applied to the motion frame along the Y axis.
Force Z	The force applied to the motion frame along the Z axis.
Moment X	The moment (torque) applied to the motion frame along the X axis.
Moment Y	The moment (torque) applied to the motion frame along the Y axis.
Moment Z	The moment (torque) applied to the motion frame along the Z axis.
Orientation Angle	For the set of axes that represents the Motion Frame's orientation, this is the rotation angle about the defined **Relative Rotation Vector**. See also About Creating and Applying Motion Frames to Custom Geometries (p. 533).
Orientation Vector X	The X-component defining the vector about which the motion frame axes is rotated. See also About Creating and Applying Motion Frames to Custom Geometries (p. 533).
Orientation Vector Y	The Y-component defining the vector about which the motion frame axes is rotated. See also About Creating and Applying Motion Frames to Custom Geometries (p. 533).
Orientation Vector Z	The Z-component defining the vector about which the motion frame axes is rotated. See also About Creating and Applying Motion Frames to Custom Geometries (p. 533).
Position X	The coordinate location of the motion frame axes' center point as specified in the X direction.
Position Y	The coordinate location of the motion frame axes' center point as specified in the Y direction.
Position Z	The coordinate location of the motion frame axes' center point as specified in the Z direction.
Velocity X	The translational velocity of the motion frame as defined in the X direction.
Velocity Y	The translational velocity of the motion frame as defined in the Y direction.
Velocity Z	The translational velocity of the motion frame as defined in the Z direction.

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

What would you like to do?

- Learn more About Customizing Properties or Curves (p. 1012)
- Add a New Custom Property on the Properties or Curves Tab (p. 1037)
- Change or Remove an Existing Custom Property on the Properties or Curves Tab (p. 1036)

See Also:

- About Properties (p. 933)
- Analyzing a Simulation (p. 931)

5.1.3. About Customizing Properties or Curves

There are many default properties provided on the Properties and Curves tabs for a simulation entity (Particles, 1-Way LBM, 1-Way Fluent, Solver, individual geometry components, individual Motion Frames, Eulerian Statistics, and any individual User Processes linked to these entities (p. 1395)) but you can also create and edit custom properties to create your own unique data sets.

Note:

Though customizing Properties also applies to Eulerian Statistics, because Eulerian Statistics is a unique User Process based only upon the Particles entity, you can also add new Particles Properties to a Eulerian Statistics User Process. See About Customizing Eulerian Statistics Properties (p. 1017) for more information.

Customizing Properties is accomplished from the upper-right corner of the Properties tab. Customizing Curves is accomplished from the upper-right corner of the Curves tab.

Tip:

To see walk-through examples of customizing a property or curve, refer to the following Tutorials:

- Workshop 03: Vibrating Screen
- Workshop 18: Mixing Analysis (Liquid Bridge and LMI)

Custom Properties or Curves Parameter Definitions

See the images and tables below to help you understand how to add and edit your own unique properties or curves.

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

Figure 5.27: Add/Edit custom (property or curve) button locations

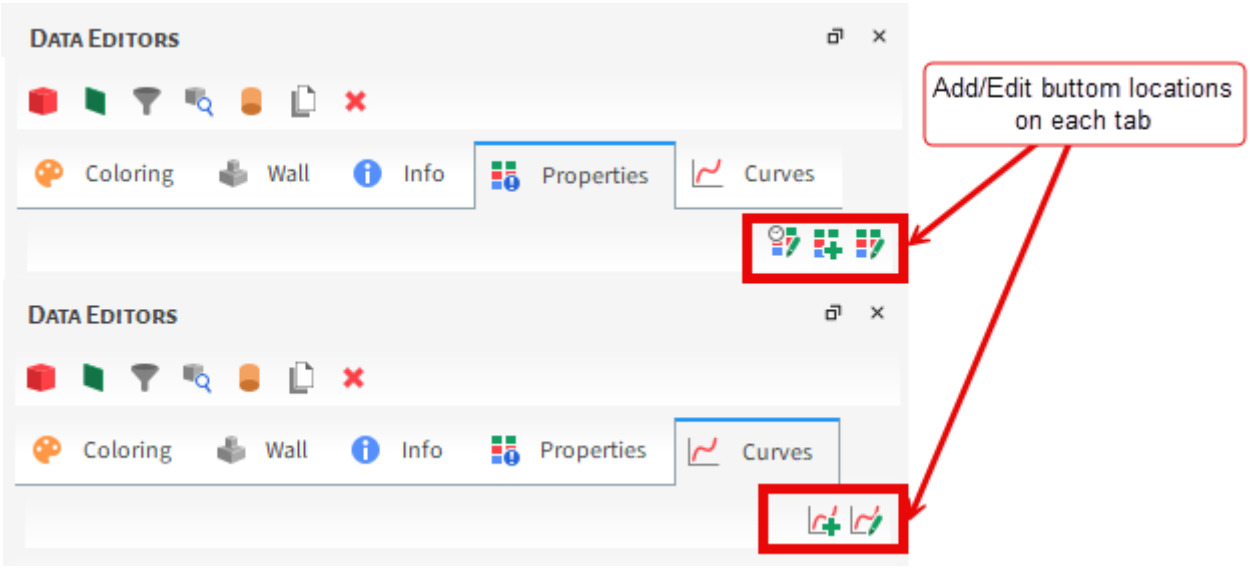
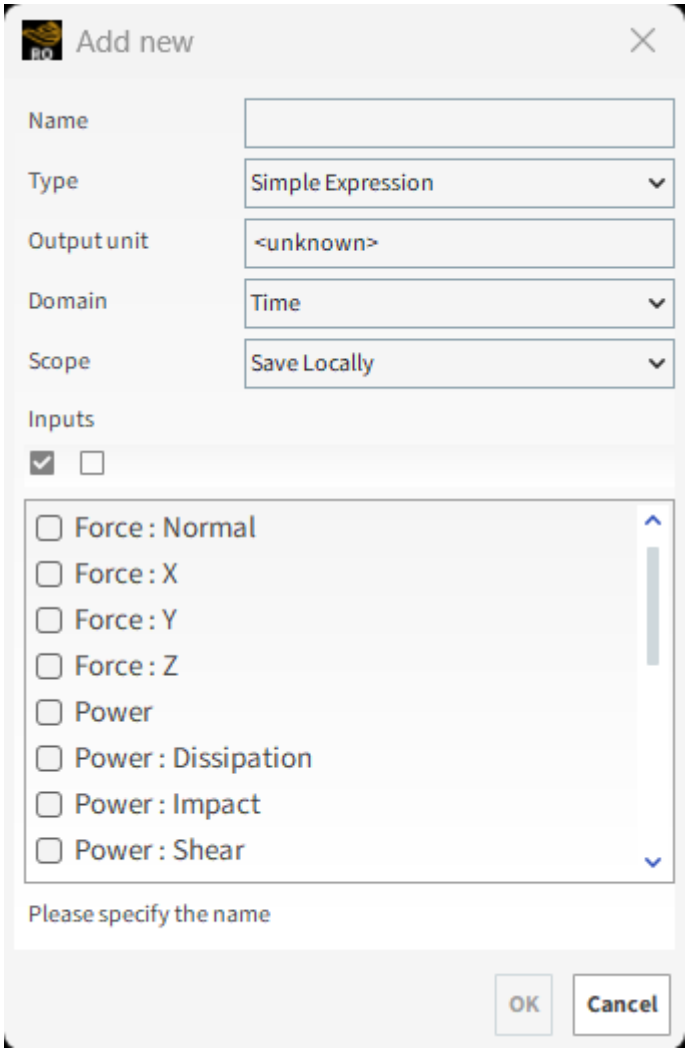


Figure 5.28: Add new dialog (property or curve)



Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

Table 1 : Add new dialog options (property or curve)

Setting	Description	Range
Name	Enables you to specify a unique identifier for the property or curve.	99 character limit
Type	Specifies how the expression will be defined. • **Simple Expression** will be defined as a mathematical function. • **Advanced Expression (script)** will be defined using Python code. **Note:** The expression is entered on the **Custom Property** or **Custom Curves** dialog that appears after you click **OK**. (See below section.)	Simple Expression; Advanced Expression (script)
Output unit	The units you want assigned to the property or curve.	<unknown> (see also Recognize Data Placeholders (p. 116)); any applicable unit defined in Rocky (see also About Setting Your Unit System (p. 238)).
Inputs	Lists all existing properties or curves available for the entity, including any custom properties or curves you have defined.	Automatically provided. (See also Use Properties and Curves (p. 931)).

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

Figure 5.29: Custom Property dialog for Simple Expressions

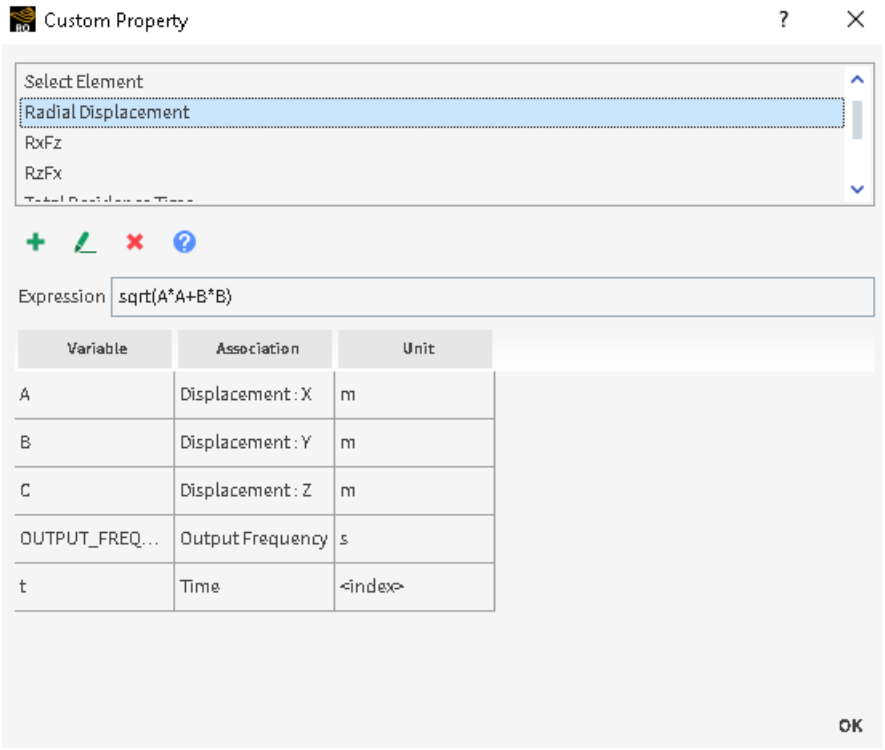


Figure 5.30: Custom Curves dialog for Advanced Expressions

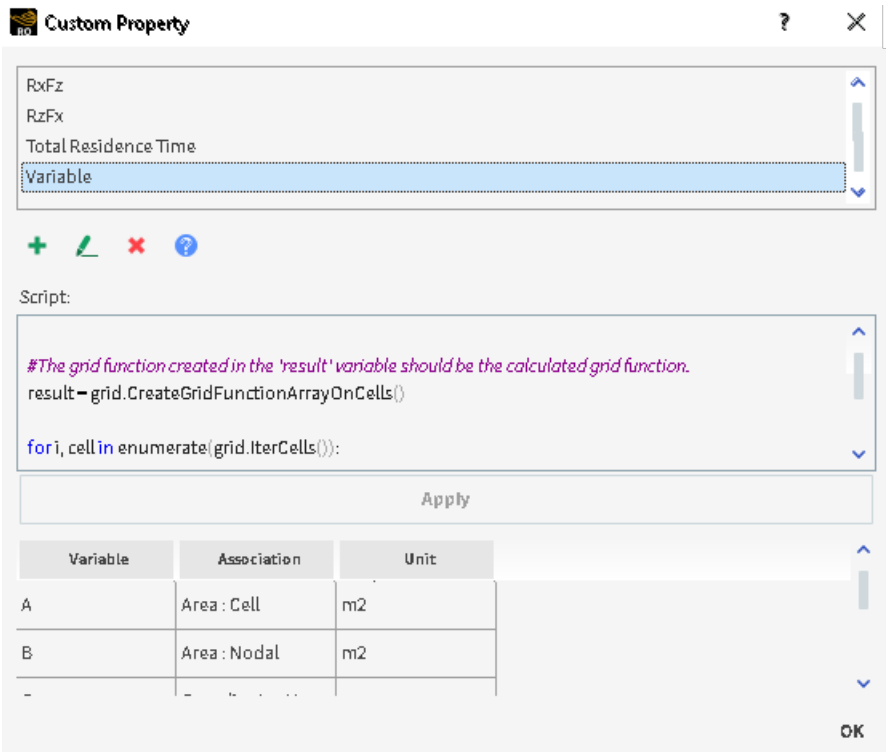


Table 2: Custom dialog options (properties and curves)

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

Setting	Description	Range
Expression	When the **Type** of the custom property or curve is **Simple Expression**, this is where you define the mathematical expression for the property or curve. **Note:** The expression in Rocky follows the syntax of the Python NumPy library.	
Script	When the **Type** of the custom property or curve is **Advanced Expression (script)**, this is where you use Python code to define the details of the property or curve. **Note:** The code you enter here is not the same as a Rocky *script* (p. 1394) which has a different structure and purpose.	See also Customize and Extend Rocky with Python (p. 294)
Variable	Column that lists the variable names available for you to use within the expression or script. Specifically: • **A** (and subsequent letters) corresponds to the properties or curves you selected from the **Inputs** list on the **Add new** dialog. • **OUTPUT_FREQUENCY** corresponds to the **Simulation** Output Frequency value you enter on the **Solver Time** tab. (See also About Solver Parameters (p. 464)). • **TIME_ELAPSED** corresponds to the amount of simulation time that was solved up to the moment since starting (or resuming) the simulation. Note: This is available for curves only. The following variables are also in the product but are not recommended for use at this time: • **TIME_SET** • **TIME_STEP**	Automatically provided
Association	Column that maps the **Variable** name to its Rocky property.	Automatically provided
Unit	Column that lists the unit used by the **Variable**.	Automatically provided

What would you like to do?

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

- Add a New Custom Property on the Properties or Curves Tab (p. 1037)
- Change or Remove an Existing Custom Property on the Properties or Curves Tab (p. 1036)

See Also:

- About Properties (p. 933)
- About Curves (p. 989)
- Analyzing a Simulation (p. 931)
- About Eulerian Statistics (p. 1226)

5.1.4. About Customizing Eulerian Statistics Properties

You can customize Eulerian Statistics Properties just like any other Property. (See also About Customizing Properties or Curves (p. 1012).) However, unlike other Properties, you may also add new Particles Properties to your default list of Eulerian Statistics Properties.

Eulerian Statistics is a unique User Process based only upon the Particles entity (see also About Eulerian Statistics (p. 1226)). The primary purpose is to analyze specific locations in a simulation, and not necessarily the individual particles that are in them. For example, you might use Eulerian Statistics to analyze the average particle mass inside each individual block to see at a glance which region of your simulation has a higher density of particles. So while the analysis uses information about the particles, it is really the individual blocks and the simulation locations they represent that are the primary focus.

Thus, unlike other User Processes, which offer default Properties shared by the simulation entity upon which they were created, Eulerian Statistics has its own set of unique Properties. (See also About Properties (p. 933).) To these default functions, you can add additional Properties from the Particle entity upon which the Eulerian Statistics User Process is based.

See the images and table below to understand how to add Particle Properties to your Eulerian Statistics User Process.

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

Figure 5.31: Add and edit properties for source entity button location

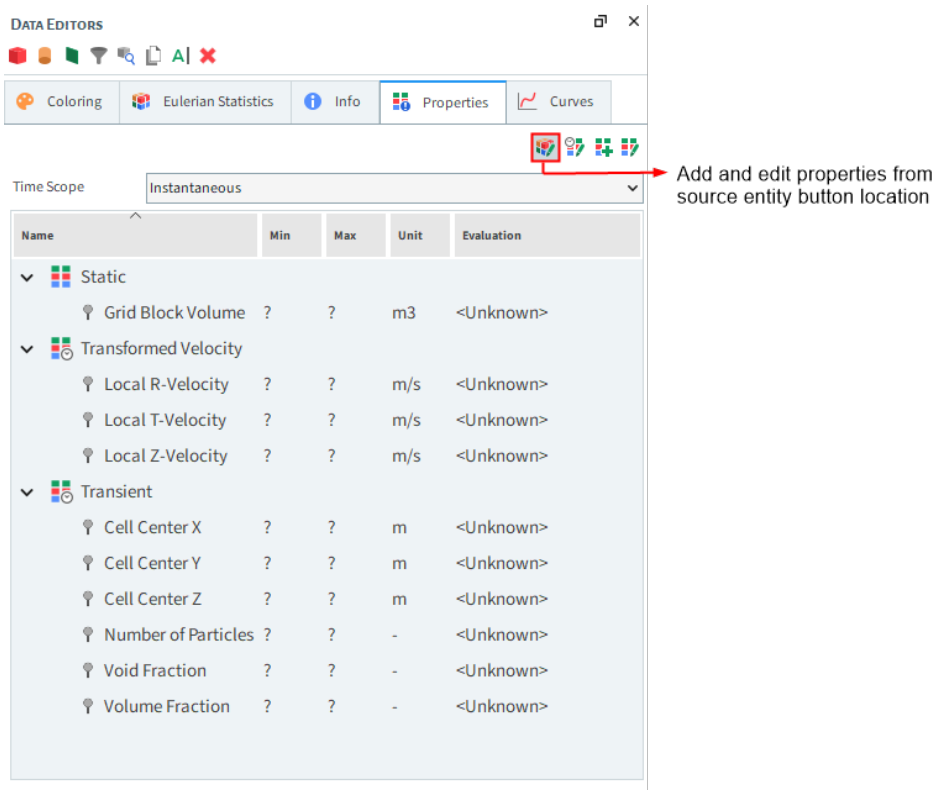


Figure 5.32: Add or edit eulerian properties dialog

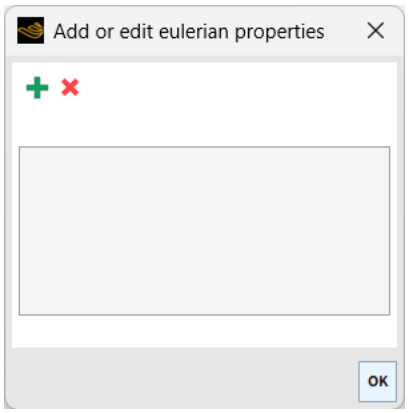


Figure 5.33: Add new property dialog



Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

Table 1 : Add new property dialog options

Setting	Description	Range
Property Source	The source entity from which you will select the property to add or edit. You can select between: • **Postprocessing Entity**: The entity upon which you created the Eulerian User Process. For example, if your Eulerian User Process is based upon a Cube User Process, this Cube will be available for selection. • **Particles**: You can select the Particles entity as a property source. • **SPH**: You can select the SPH entity as a property source.	Postprocessing Entity; Particles; SPH
Operation	The operation that will be applied to the Property(s) selected. Specifically: • **Average** is the arithmetic mean, defined as the sum of the numerical values of each and every observation divided by the total number of observations. Suppose Particles is the selected source entity. If the bin contains N particles and the property value of each particle is a_n , the average A is computed as: $A=\frac{1}{N}\sum_{i=1}^{N}a_i=\frac{a_1+a_2+...+a_n}{N}$ • **Min** returns the sample minimum, also called the smallest observation, i.e., the values of the least elements of a sample of n elements. Suppose Particles is the selected source entity. If the bin contains N particles and the property value of each particle is a_n, the min returns the first order statistics, $X_{(1)}$ according to the equation: $A=X_{(1)}=\min\left(X_1,X_2,...,X_n\right)$ • **Max** returns the sample maximum, also called the largest observation, i.e., the values of the greatest elements of a sample of n. Suppose Particles is the selected source entity. If the bin contains N particles and the property value of each particle is a_n, the max returns the largest order statistics, $X_{(n)}$ according to the equation: $A=X_{(n)}=\max\left(X_1,X_2,...,X_n\right)$	Average; Min; Max; Sum; Volume Fraction Average; Weighted Average; Standard Deviation

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

Setting	Description	Range
	• **Sum** returns the addition of a sequence of N values. Suppose Particles is the selected source entity. If the bin contains N particles and the property value of each particle is a_n, the sum is computed as: $$A=\sum_{n=1}^{N} a_n = a_1 + a_2 + ... + a_n$$ • **Volume Fraction Average** returns the summation of the desired property for each particle, a_n, weighted by the volume of the particle or, V_n, divided by the volume of the bin, V_b, as shown in the following equation: $$A=\frac{\sum_{n=1}^{N} a_n V_n}{V_b} = \frac{a_1 V_1 + a_2 V_2 + ... + a_n V_n}{V_b}$$ The same is valid for SPH as source entity. • **Weighted Average** returns the weighted arithmetic mean. Different from the ordinary arithmetic mean, some points here contribute more than others for the final average, i.e., each point has a certain weight. Therefore, data elements with a high weight contribute more to the weighted mean than do elements with a low weight. If the bin contains N particles or SPH elements and the property value of each particle is a_n, and the weights are ω_n, the weighted mean is computed as: $$A=\frac{\sum_{n=1}^{N} a_n \omega_n}{\sum_{n=1}^{N} \omega_n} = \frac{a_1 \omega_1 + a_2 \omega_2 + ... + a_n \omega_n}{\omega_1 + \omega_2 + ... + \omega_n}$$ The same is valid for SPH as source entity. • **Standard Deviation** is used to quantify the amount of variation or dispersion within a set of property values. A low standard deviation indicates that the analyzed property tends to be close to the mean amongst the particles in the bin, while a high standard deviation indicates that the data points are spread out over a wider range of values. If the bin contains N particles and the property value of each particle is a_n, the standard variation is computed as: $$\sigma=\sqrt{\frac{1}{N-1}\sum_{n=1}^{N}\left(a_n-\mu\right)^2}$$	

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

Setting	Description	Range
	where the average is computed as: $\mu=\frac{1}{N-1}\sum_{n=1}^{N}a_n$ The same is valid for SPH as source entity.	
Property	Lists all existing properties available for the source entity, including any custom properties you have defined.	Automatically provided. (See also About Properties (p. 933).)
Weight Property	When **Weighted Average** is chosen from the **Operation** list, this lists all existing properties available for the source entity, including any custom properties you have defined.	Automatically provided. (See also About Properties (p. 933).)

What would you like to do?`

- Add a Property to a Eulerian Statistics User Process (p. 1037)
- Remove a Property from an Eulerian Statistics User Process (p. 1038)

See Also:

- Filter Views and Data with User Processes (p. 1174)
- About Properties (p. 933)
- About Eulerian Statistics (p. 1226)

5.1.5. About SPH Eulerian Solution

You can customize the SPH Eulerian Solution Properties just like any other Property. (See also About Customizing Properties or Curves (p. 1012).)

Eulerian Solution allows you to analyze specific locations in a simulation where there are no SPH elements in a specific time step. For example, you might use Eulerian Solution to analyze the average density of each individual cell see at a glance which region of your simulation has a higher fluid density. So while the analysis uses information about the SPH elements, it is really the individual cells and the simulation locations they represent that are the primary focus.

The Eulerian Solution option has its own set of unique Properties. (See also About Properties (p. 933).) To these default functions, you can add additional Custom Properties.

See the image and table below to understand what Properties are available when using the Eulerian Solution.

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

Figure 5.34: Properties available in the SPH Eulerian Solution

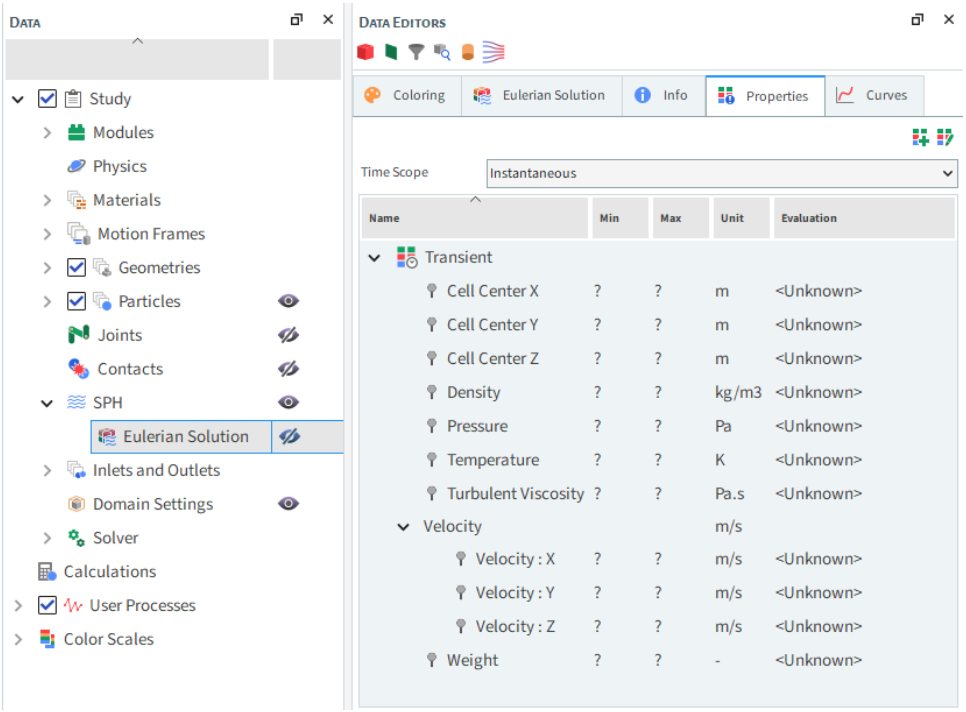


Table 1 : Properties available in the SPH Eulerian Solution

Property Name	Description
Cell Center X	Eulerian bin coordinates in the X direction. Useful for analyzing properties that can vary spatially, you can create a Cross Plot of a certain property over the bin coordinates in the X direction.
Cell Center Y	Eulerian bin coordinates in the Y direction. Useful for analyzing properties that can vary spatially, you can create a Cross Plot of a certain property over the bin coordinates in the Y direction.
Cell Center Z	Eulerian bin coordinates in the Z direction. Useful for analyzing properties that can vary spatially, you can create a Cross Plot of a certain property over the bin coordinates in the Z direction.
Density	The average density of each Eulerian Solution cell
Pressure	The average pressure of each Eulerian Solution cell
Turbulent Viscosity (only available if a turbulence model is enabled)	The average turbulent viscosity of each Eulerian Solution cell
Velocity: X	The average X direction velocity of each Eulerian Solution cell
Velocity: Y	The average Y direction velocity of each Eulerian Solution cell
Velocity: Z	The average Z direction velocity of each Eulerian Solution cell
Weight	Property that indicates from 0 to 1 how much fluid is present in a cell (0 no fluid, 1 full of fluid)

See Also:

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

- Filter Views and Data with User Processes (p. 1174)
- About Properties (p. 933)
- About Eulerian Statistics (p. 1226)

5.1.6. About Adding and Editing Time Statistics Properties

A Time Statistics Property enables you to limit a Property by a particular time range and operation that you specify. That new property definition becomes its own unique Property, which you can then use just like any other Property for further analysis. This method is useful when you need to focus your analysis upon only the average (max, min, or sum) values for a Property calculated during a certain time period of your simulation—after steady state has been reached or once particles start leaving the simulation boundaries, for example.

Tip:

To see a walk-through example of adding and using a Particle Time Statistics Property, review Tutorial - Tablet Coater in the *Rocky Tutorial Guide*.

Applying Time Statistics Properties

Time statistics are available for all Properties (see also About Properties (p. 933)) except those accessible via the main **Particles**, the **1-Way Fluent** CFD Coupling method, and **Contacts** entities (p. 1395). This includes support for all of the following entities:

- Individual Particle set Properties. (See also About Intra-Particle Collision Statistics (p. 324).)
- Eulerian Statistics User Process Properties. (See also About Eulerian Statistics (p. 1226).)
- Individual Geometry component Properties.
- 1-Way LBM Properties. (See also About Using the 1-Way LBM Method (p. 771).)

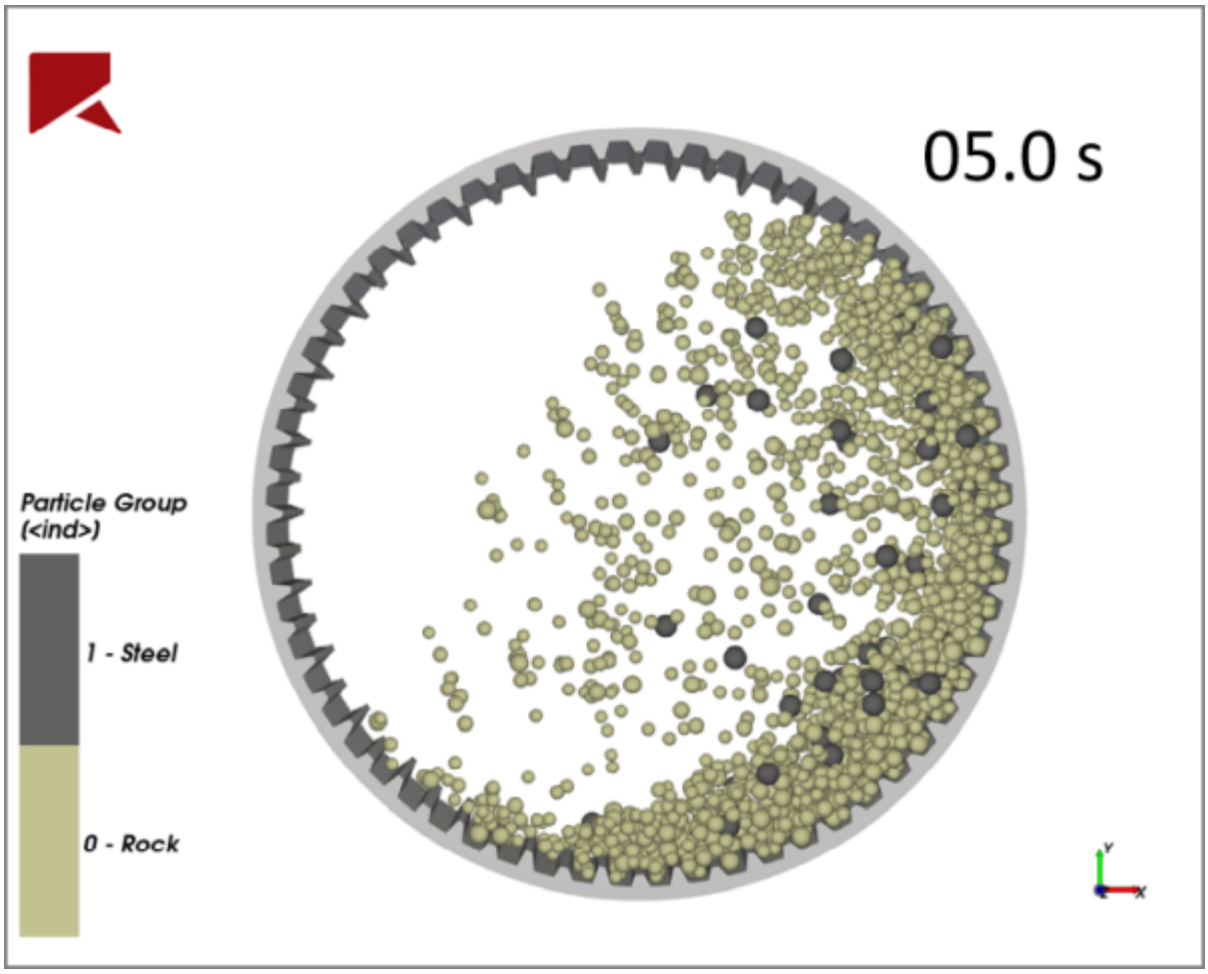
Once you add a Time Statistics Property for one of the above supported entities, it will be available for all entities of the same type.

Time Statistics Properties and Eulerian Statistics

With regular Properties, the results by default only take into account data calculated for the current time step (Figure 5.36: The same mill and time step as above, but showing one Property (standard Property) (p. 1025)). By using by Eulerian Statistics User Process, for example, and then customizing the Properties (see also About Customizing Eulerian Statistics Properties (p. 1017)), you could also show an operation (average, for example) for that same Property within the individual Eulerian blocks, but still for only one time step at a time (Figure 5.37: The same mill and time step as Figure 2 but with Eulerian Statistics applied showing the average values of a Property within each Eulerian block (custom Property) (p. 1026)). With Time Statistics Properties, you can now apply an operation over a specific time range (Figure 5.38: The same mill and Eulerian Statistics applied as above, but now those values are averaged over a set range of time (Time Statistics Property) (p. 1027)).

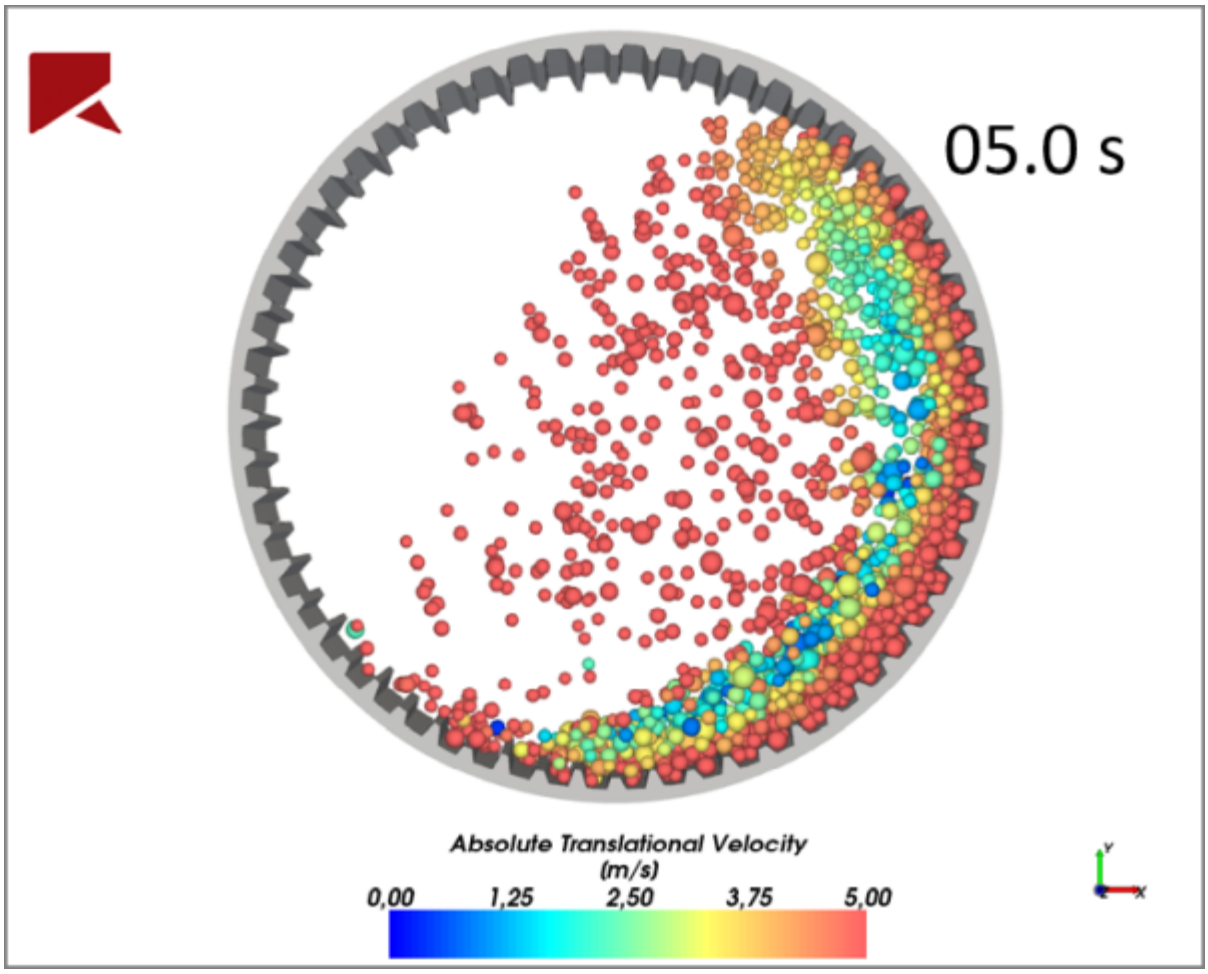
Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

Figure 5.35: A rotating mill at a single time step



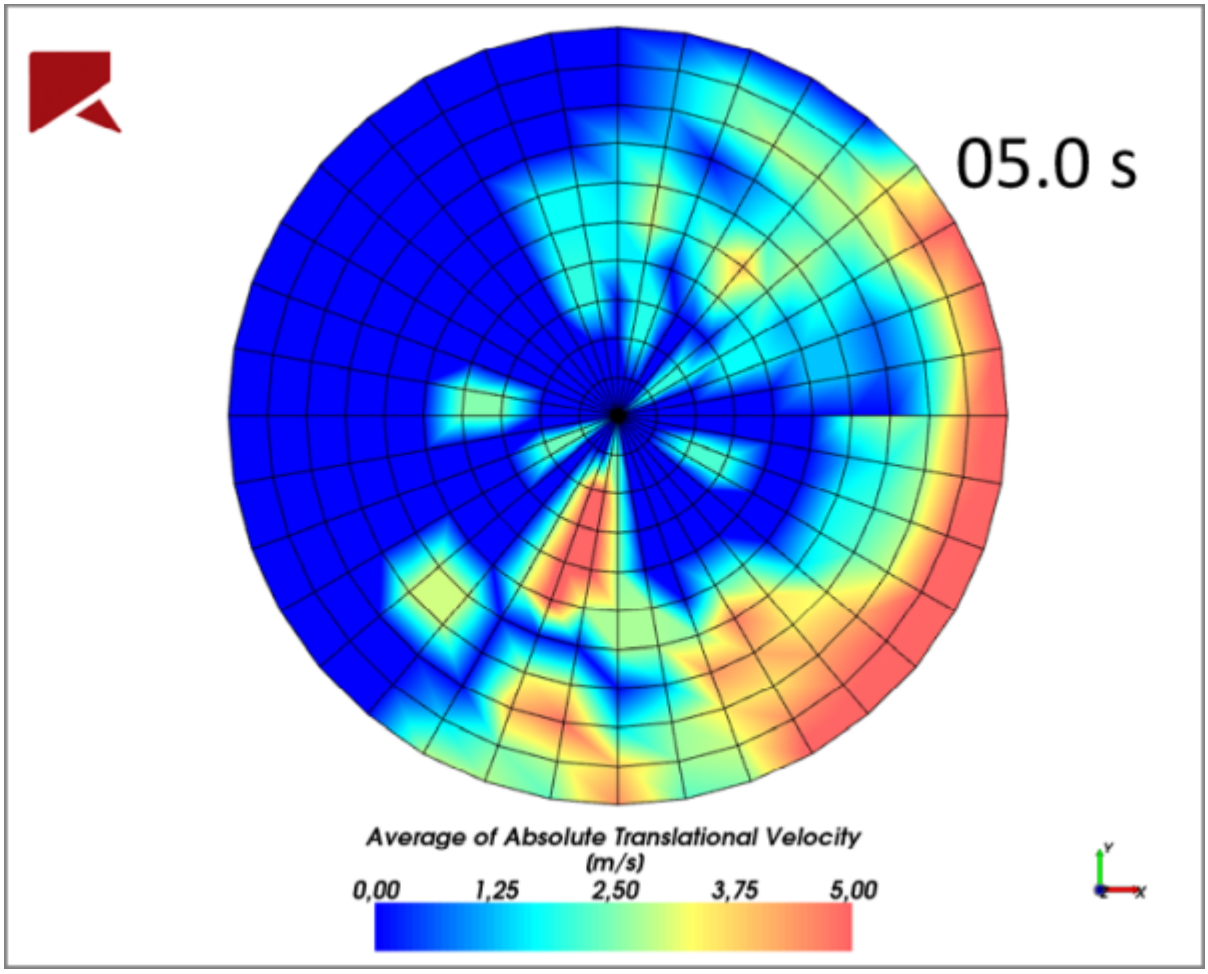
Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information
of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

Figure 5.36: The same mill and time step as above, but showing one Property (standard Property)



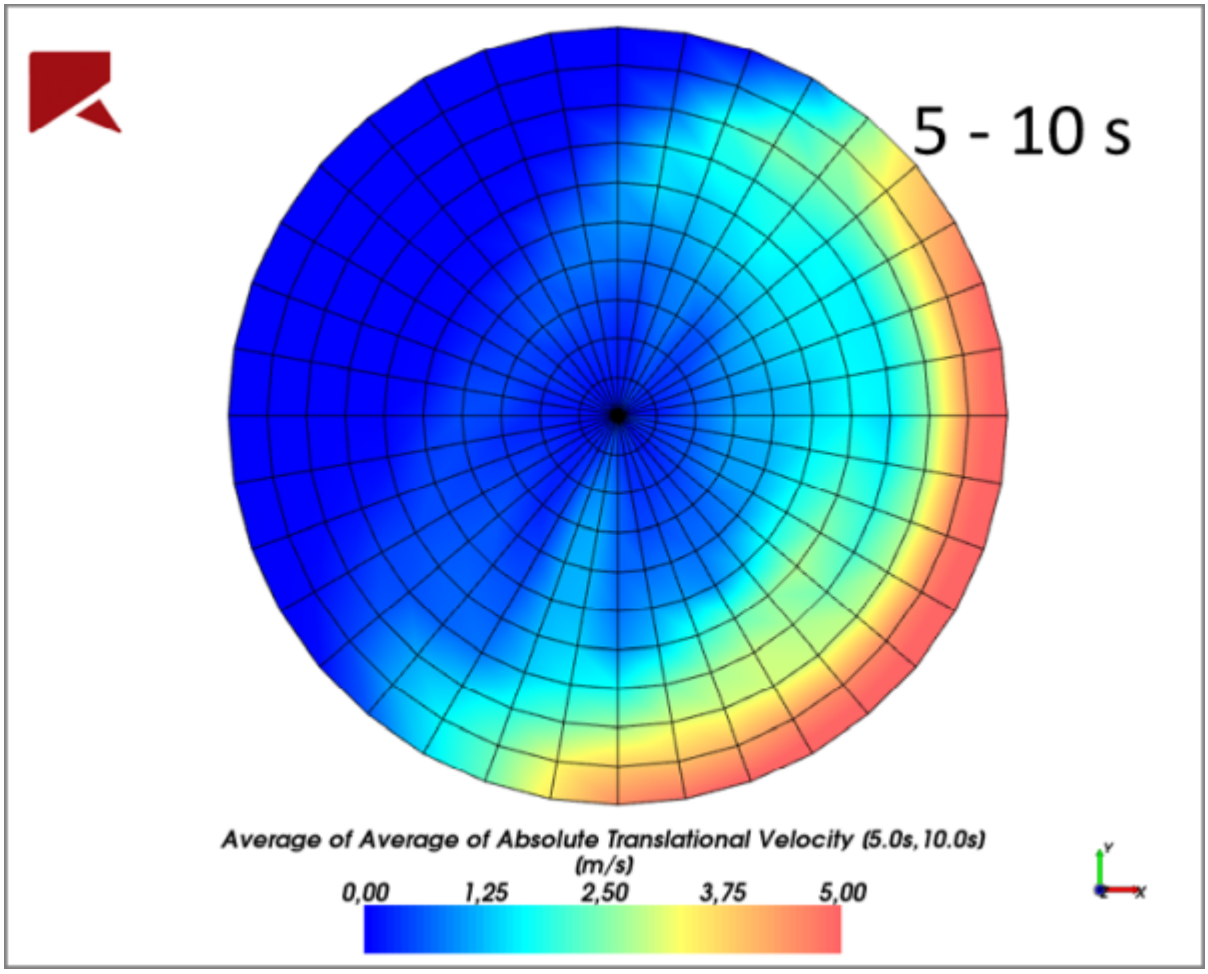
Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information
of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

Figure 5.37: The same mill and time step as Figure 2 but with Eulerian Statistics applied showing the average values of a Property within each Eulerian block (custom Property)



Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

Figure 5.38: The same mill and Eulerian Statistics applied as above, but now those values are averaged over a set range of time (Time Statistics Property)



Time Statistics Properties Parameter Definitions

Like Customizing a Property (see About Customizing Properties or Curves (p. 1012)), adding and editing Time Statistics Properties is accomplished from the upper-right corner of the Properties tab. See the images and tables below to help you understand how to add and edit Time Statistics Properties.

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

Figure 5.39: Add and edit time statistics properties button location

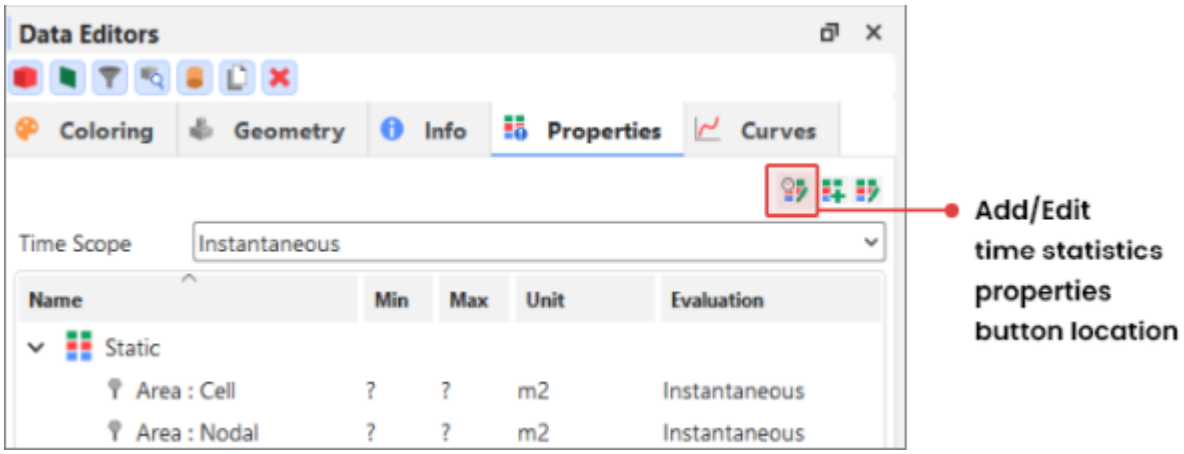
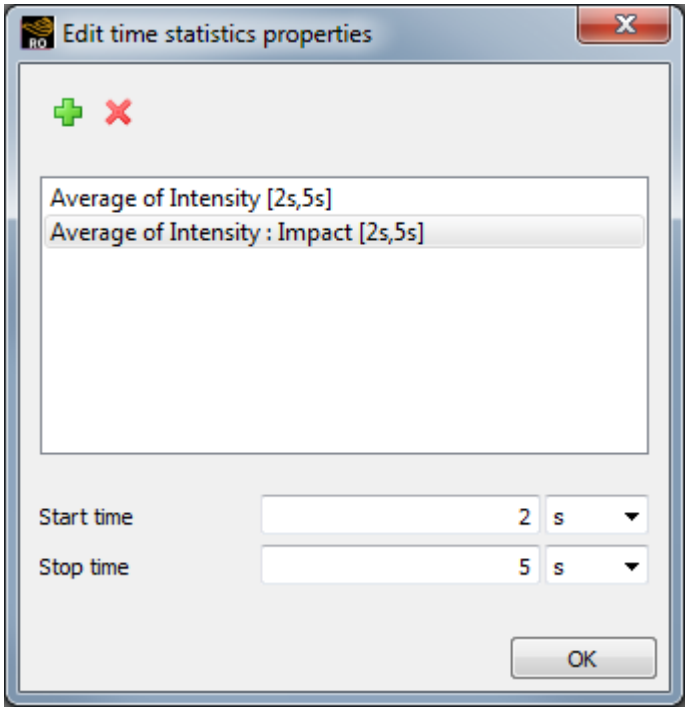
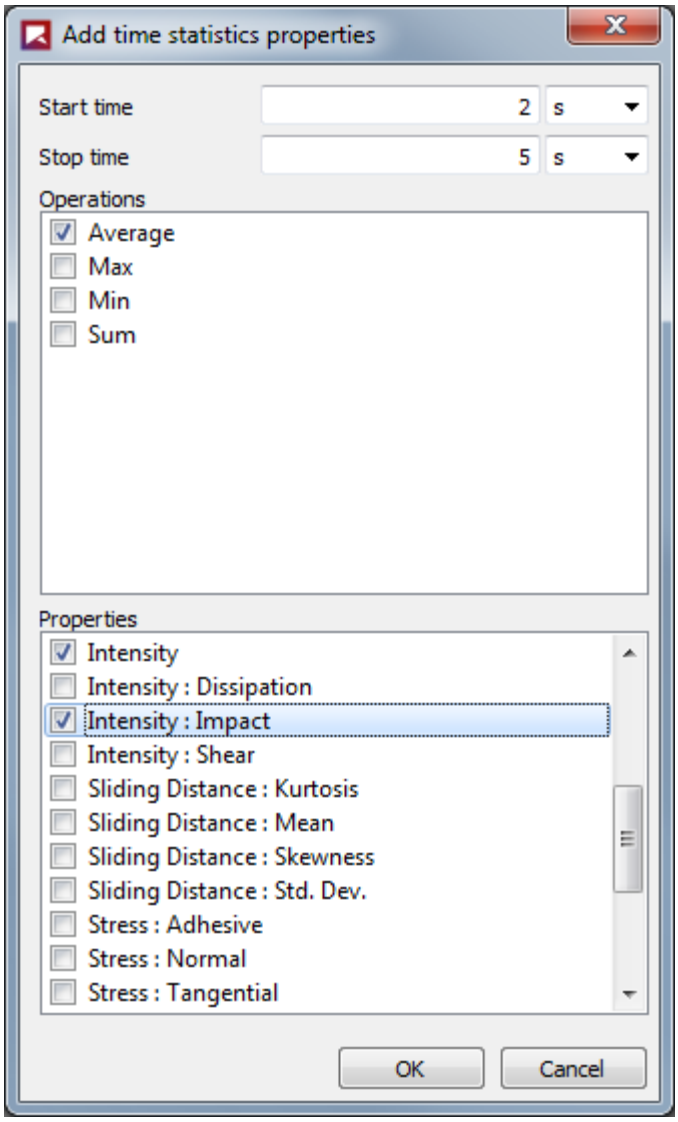


Figure 5.40: Edit time statistics properties dialog showing two new Properties



Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

Figure 5.41: Add time statistics properties dialog



Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

Figure 5.42: Two newly added Time Analysis Properties in the Data Editors panel

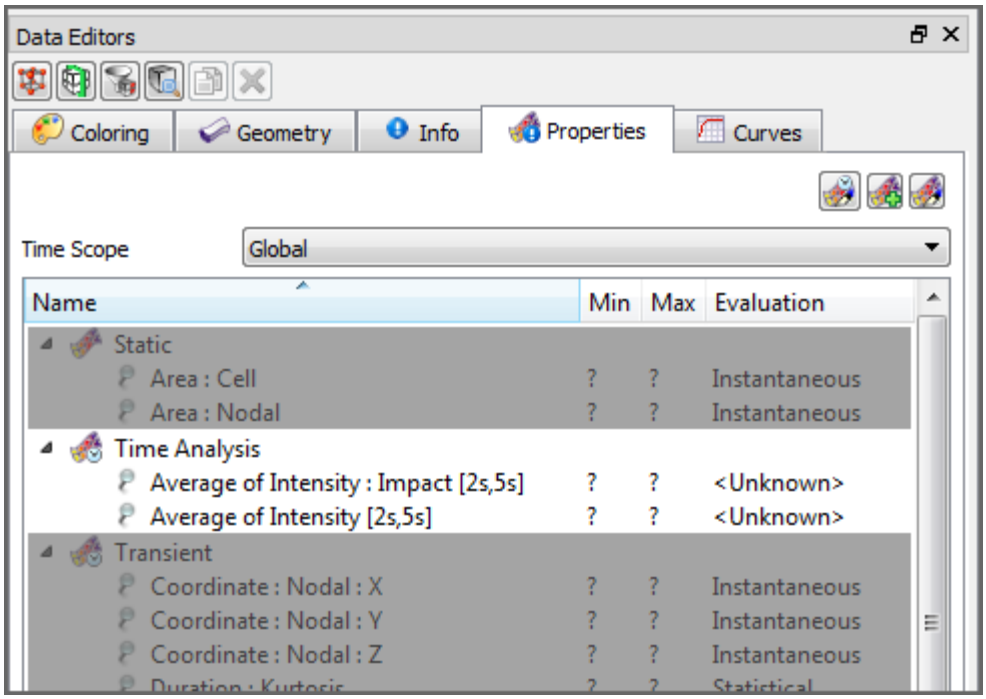


Table 1: Edit time statistics property dialog options

Setting	Description	Range
Start Time	The amount of time into your simulation that you want calculations to start being included in the operation selected for the Time Statistics Property. **Note:** Neither Time values are editable until you add a Time Statistics Property. To accomplish this, click the **Add** button. (See also Add a Time Statistics Property (p. 1039).)	Positive values

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

Setting	Description	Range
Stop Time	The amount of time into your simulation that you want calculations to stop being included in the operation selected for the Time Statistics Property. **Tip:** If you want your range to extend to the end of your simulation, enter a very large number, such as 1000. **Note:** Neither Time values are editable until you add a Time Statistics Property. To accomplish this, click the **Add** button. (See also Add a Time Statistics Property (p. 1039).)	Positive values

Table 2: Add time statistics property dialog options

Setting	Description	Range
Start Time	The amount of time into your simulation that you want calculations to start being included in the operation selected for the Time Statistics Property. Between this and the **Stop Time** value defines the time range for the new Time Statistics Property.	Positive values
Stop Time	The amount of time into your simulation that you want calculations to stop being included in the operation selected for the Time Statistics Property. Between the **Start Time** and this value defines the time range for the new Time Statistics Property. **Tip:** If you want your range to extend to the end of your simulation, enter a very large number, such as 1000.	Positive values
Operations	Enables you to choose one or more operations to apply to the Property data within the time range you selected. One unique Time Statistics Property will be created per **Operation** and **Property** selected. Specifically: • **Average** is the temporal mean, defined as the arithmetic mean of a series of values over a time period. Since the sampling times are equal (simulation output frequency is constant), it can be computed as the sum of the variable values over the desired time period divided by the number of values. For a simulation with output frequency of Δt, in which the time range for the averaging is comprehended between t_i and t_f and contains	Average; Min; Max; Sum

Setting	Description	Range
	N outputs and the property value of the evaluate bin at each time is a_n, the temporal average A is computed as: $A=\frac{1}{N}\sum_{n=1}^{N}a_n=\frac{a_1+a_2+...+a_n}{N}$ where N is computed as: $N=\frac{t_f-t_i}{\Delta t}$ • **Min** returns the sample minimum, also called the smallest observation. For a simulation with output frequency of Δt, in which the time range for the averaging is comprehended between t_i and t_f and contains N outputs and the property value of the evaluated bin at each time is a_n, the min returns the first order statistics, $X_{(1)}$ according to the equation: $A=X_{(1)}=\min\left(X_1,X_2,...,X_n\right)$ • **Max** returns the sample maximum, also called the largest observation. For a simulation with output frequency of Δt, in which the time range for the averaging is comprehended between t_i and t_f and contains N outputs and the property value of the evaluated bin at each time is a_n, the max returns the largest order statistics, $X_{(n)}$ according to the equation: $A=X_{(n)}=\max\left(X_1,X_2,...,X_n\right)$ • **Sum** returns the addition of a sequence of N values. For a simulation with output frequency of Δt, in which the time range for the averaging is comprehended between t_i and t_f and contains N outputs and the property value of the evaluated bin at each time is a_n, the sum is computed as: $A=\sum_{n=1}^{N}a_n=a_1+a_2+...+a_n$ where N is computed as: $N=\frac{t_f-t_i}{\Delta t}$	
Property	Enables you to choose one or more Properties to which you can apply the selected **Operations** over the time range you selected. One unique Time Statistics Property with the selected time range will be created per **Property** and **Operation** selected.	List provided automatically

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

What would you like to do?`

- Add a Time Statistics Property (p. 1039)
- Edit or Remove a Time Statistics Property (p. 1039)
- I cannot find the Output Frequency parameter (p. 1336)

See Also:

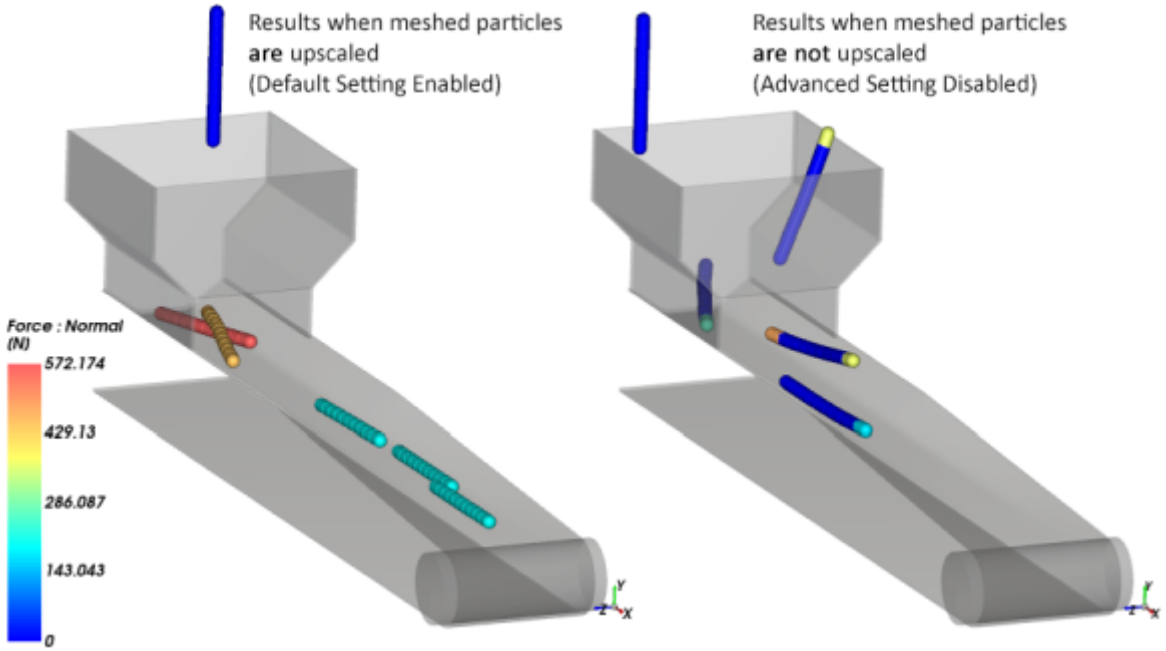
- About Properties (p. 933)
- About Eulerian Statistics (p. 1226)

5.1.7. About Meshed Particles Upscaling

The concept of particle upscaling (p. 1396) refers to the way Rocky treats post-processing data and statistics from individual elements (p. 1390) of a Multiple Element (meshed (p. 1392)) particle. While the Rocky solver always computes multi-element properties per individual element, upscaling determines whether or not the application extends those individual properties to the whole particle itself.

In general, if a meshed particle is upscaled, which is the default setting (as shown below on the left), post-processing results will be provided for the whole particle. If a meshed particle is not upscaled, meaning the setting was disabled (as shown below on theright) post-processing results will be provided for each individual Element making up the particle.

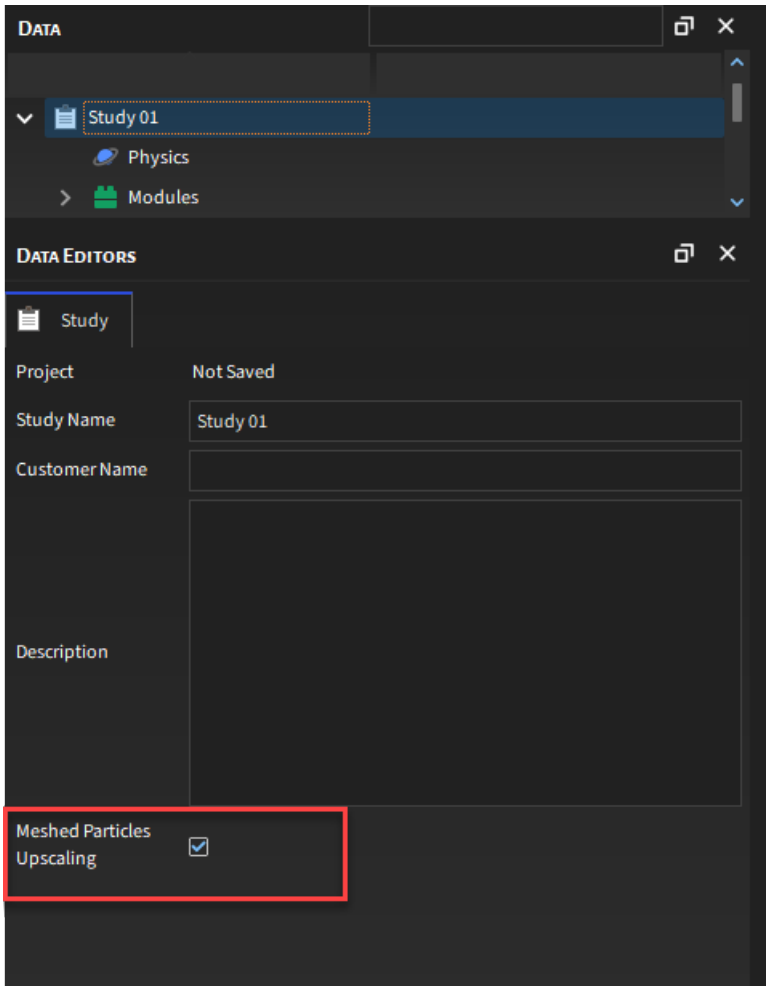
Figure 5.43: Example results when meshed particle upscaling is enabled (left) and disabled (right)



How this is applied in your simulation is determined by the **Meshed Particles Upscaling** checkbox in the **Study** parameter on the Data Editors (as shown below). (See also About Setting Global Preferences (p. 227).)

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

Figure 5.44: Meshed Particles Upscaling, Study parameter



Note:

As this setting applies only to post-processing analyses, you can adjust this parameter at any time during your simulation. However, you will need to close and reopen your Rocky project for you to see the change applied to your results.

About Upscaled Particles (Default Setting)

By default, the **Meshed Particles Upscaling** checkbox is enabled. In these cases, any Multi-Element particle used in the simulation is an upscaled one and *most* post-processing properties and curves will be provided for the particle as a whole (Figure 5.43: Example results when meshed particle upscaling is enabled (left) and disabled (right) (p. 1033), left).

There are some known exceptions to this rule, however. These include the Particles and Particle set properties and curves listed in the tables below. In these cases, even when the simulation has meshed particles set to be upscaled, the post-processing data might still be provided for the individual Elements (upscaling is ignored — Tables 1 and 2) or will not be provided at all (not listed in Properties tab — Table 3).

Table 1: Upscaling exceptions for Particle Set Properties (Per Set Only)

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

Effects of upscaling are ignored for this Property	How Collected	See Also
(All Intra-particle properties)	Intra-particle Collision Statistics	About Intra-Particle Collision Statistics (p. 324)

Table 2: Upscaling exceptions for Particle Curves (Simulation Wide)

Effects of upscaling are ignored for this Curve	How Collected	See Also
(All Inter-group curves)	Inter-group Collision Statistics	About Inter-group Collision Statistics (p. 319)

Table 3: Upscaling exceptions for Particles Properties (Simulation Wide)

Upscaling results in this Property not being listed on the Properties tab	How Collected	See Also
Duration	Inter-particle Collision Statistics	About Inter-particle Collision Statistics (p. 331)
Velocity : Impact	Inter-particle Collision Statistics	About Inter-particle Collision Statistics (p. 331)
Moment of Inertia	Automatic	About Properties (p. 933)
Orientation Angle	Automatic	About Properties (p. 933)
Orientation Vector X	Automatic	About Properties (p. 933)
Orientation Vector Y	Automatic	About Properties (p. 933)
Orientation Vector Z	Automatic	About Properties (p. 933)
Particle Equivalent Diameter	Automatic	About Properties (p. 933)
Particle Size	Automatic	About Properties (p. 933)

About Particles with Upscaling Disabled

If you decide to clear the **Meshed Particles Upscaling** checkbox (Figure 5.44: Meshed Particles Upscaling, Study parameter (p. 1034)), any Multi-Element particle used in the simulation will have its properties or curves provided per individual Element (Figure 5.43: Example results when meshed particle upscaling is enabled (left) and disabled (right) (p. 1033), right).

Notes:

- Because the properties and curves listed above in Tables 1 and 2 are ignoring the effects of upscaling, disabling upscaling in these cases results in the same outcome as if it was enabled: for Multi-Element particles, the properties and curves are provided per individual Element only.
- In this version of Rocky, disabling upscaling is the only way to obtain the properties shown above in Table 3.
- Upscaling is not compatible with deformable particles, to use this feature upscaling needs to be disabled.

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

See Also:

- About Setting Global Preferences (p. 227)
- About Properties (p. 933)
- About Intra-Particle Collision Statistics (p. 324)
- About Inter-particle Collision Statistics (p. 331)
- About Inter-group Collision Statistics (p. 319)
- Understand Data Collection in Rocky (p. 306)

5.1.8. Change or Remove an Existing Custom Property on the Properties or Curves Tab

Note:

This particular process does not apply to any Particles Properties added to a Eulerian Statistics User Process. See About Customizing Eulerian Statistics Properties (p. 1017) for more information.

1. From the **Data** panel, select the entity containing the property you want to change or remove. This can include a conveyor or imported geometry component, Particles, 1-Way LBM, Solver, or individual Motion Frame.
2. From the **Data Editors** panel, select either the **Properties** or **Curves** tab containing the custom property you want to change or remove.
3. From the upper right corner of the tab, click the **Edit custom…** button.
4. From the **Custom…** dialog, select the name of the property you want under **Select Element**.
5. Do any or all of the following:
   - To change the **Name**, **Output unit**, or **Inputs** used by the property, click the **Edit….** button.
   - To change the expression listed in either the **Expression** or **Script** boxes, make your changes directly within the dialog.
   - To remove the function, click the **Remove custom…** button.
6. Click **OK**.

See Also:

- Add a New Custom Property on the Properties or Curves Tab (p. 1037)
- About Customizing Properties or Curves (p. 1012)

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

5.1.9. Add a New Custom Property on the Properties or Curves Tab

Note:

This particular process does not apply to any Particles Properties added to a Eulerian Statistics User Process. See About Customizing Eulerian Statistics Properties (p. 1017) for more information.

1. From the **Data** panel, select the entity to which you want to add a custom property. This can include a conveyor or imported geometry component, Particles, 1-Way LBM, 1-Way Fluent, Solver, individual Motion Frame, or any individual User Process linked to these entities.
2. From the **Data Editors** panel, select one of the following:
   - The **Properties** tab, to create a custom function applying to each individual element in a structure (e.g., particle, geometry triangle, or air flow node).
   - The **Curves** tab, to create a custom property applying to the entire entity as a whole.
3. From the upper right corner of the tab, click the **Add new custom…** button.
4. From the **Add new** dialog, enter the information you want, and then click **OK**. Tips: To select all inputs, click the **Check All** button. To clear all inputs, click the **Uncheck All** button.
5. From the **Custom…** dialog, define your expression in either the **Expression** or **Script** box shown using the variables provided in the dialog's table.
6. Click **OK**. The new custom property appears in the functions list sorted by **Name**. (See also Name Components to Affect Sort Order (p. 112).)

See Also:

- About Customizing Properties or Curves (p. 1012)
- Change or Remove an Existing Custom Property on the Properties or Curves Tab (p. 1036)

5.1.10. Add a Property to an Eulerian Statistics User Process

1. From the **Data** panel, under **User Processes**, select the **Eulerian Statistics** User Process you want. (See also Divide a Cube or Cylinder User Process into Many Analysis Blocks (p. 1226).)
2. From the **Data Editors** panel, select the **Properties** tab.
3. From the upper-right corner of the tab, click the **Add and edit properties for source entity** button.
4. From the **Add or edit eulerian properties** dialog, click the **Add** button.
5. From the **Add new property** dialog, select the options you want, and then click **Add Property**. The new function appears on the **Properties** tab.
6. Repeat step 5 for each new function you want to add.

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

7. Click **Close**. The function(s) you added appear in the **Add or edit eulerian properties** dialog.
8. Click **OK**.

Note:

When saving a simulation as a restart, existing contacts and forces are not stored. When the new, restarted simulation begins, new contacts will be identified and the related forces calculated, therefore, the initial results of a restart case may differ from those obtained at the saving point in the original case. This is specially true for history-dependent contact models, such as the Hysteretic Linear Spring (HLS).

See Also:

- Remove a Property from an Eulerian Statistics User Process (p. 1038)
- About Customizing Eulerian Statistics Properties (p. 1017)
- About Eulerian Statistics (p. 1226)

5.1.11. Remove a Property from an Eulerian Statistics User Process

---

Note:

This procedure applies only to Properties that you have added to an Eulerian Statistics User Process. (See also Add a Property to an Eulerian Statistics User Process (p. 1037).) You may not remove the Properties provided by default with Rocky.

---

1. From the **Data** panel, under **User Processes**, select the **Eulerian Statistics** User Process containing the Property you want to remove.
2. From the **Data Editors** panel, select the **Properties** tab.
3. From the upper-right corner of the tab, click the **Add and edit properties for source entity** button.
4. From the **Add or edit eulerian properties** dialog, select the Property you want from the list, and then click the **Remove** button.
5. Click **OK** to close the dialog. Your changes are reflected in the Properties list for the Eulerian Statistics User Process.

See Also:

- About Customizing Eulerian Statistics Properties (p. 1017)
- About Eulerian Statistics (p. 1226)

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

5.1.12. Add a Time Statistics Property

Note:

This particular process does not apply to the main Particles or 1-Way Fluent entities (p. 1395), nor any User Process created from the Particles entity, with the exception of Eulerian Statistics. (See also About Eulerian Statistics (p. 1226).)

1. From the **Data** panel, select the entity to which you want to add a Time Statistics Property. This can include any individual geometry component, individual Particle set, 1-Way LBM, Eulerian Statistics User Process, or any other User Process created from a non-Particles entity.
2. From the **Data Editors** panel, select the **Properties** tab.
3. From the upper right corner of the tab, click the **Add and edit time statistics properties** button.
4. From the **Edit time statistics properties** dialog, click the **Add** button.
5. From the **Add time statistics properties** dialog, do all of the following:
   a. Enter the time range you want to focus upon in the **Start time** and **Stop time** boxes.
   b. From the **Operations** list, select one or more operations you want to apply to the selected Property across the time range selected.
   c. From the **Properties** list, select one or more Properties.
   d. Click **OK**. One unique Time Statistics Property per **Property** and **Operations** combination appears in the **Edit time statistics properties** dialog.
6. Click **OK**. For all entities of the same type you initially selected, the new Property(s) appear on the **Properties** tab of the **Data Editors** panel under **Time Analysis**.

See Also:

- Edit or Remove a Time Statistics Property (p. 1039)
- About Adding and Editing Time Statistics Properties (p. 1023)

5.1.13. Edit or Remove a Time Statistics Property

Note:

This particular process does not apply to the main Particles, 1-Way Fluent, nor Contacts entities (p. 1395), nor any User Process created from the Particles entity, with the exception of Eulerian Statistics. (See also About Eulerian Statistics (p. 1226).)

1. From the **Data** panel, select the entity for which you want to edit a Time Statistics Property that you have already added. (See also Add a Time Statistics Property (p. 1039).) This can include any individual geometry component, individual Particle set, 1-Way LBM, Eulerian Statistics User Process, or any other User Process created from a non-Particles entity.

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

2. From the **Data Editors** panel, select the **Properties** tab.
3. From the upper right corner of the tab, click the **Edit time statistics properties** button.
4. From the **Edit time statistics properties** dialog, select the Time Statistics Property you want to edit or remove from the list, and then do one of the following:
   - To edit it, change the **Start time** or **Stop time** values.
   - To remove it, click the **Remove** button. Your changes are reflected in the dialog.
5. Click **OK** to close the dialog. Your changes are reflected in the Properties list under **Time Analysis**.

See Also:

- Add a Time Statistics Property (p. 1039)
- About Adding and Editing Time Statistics Properties (p. 1023)

5.2. Create and Modify a 3D View Window

A 3D View is a window in the Rocky Workspace (p. 1396) that enables you to visualize the various components that make up your simulation.

After you set up your 3D View windows, you can choose to reuse the zoom, rotation, and pan settings in another window by saving and applying a Custom Camera Preset. You can also reuse your 3D View window settings within another project by exporting the project context, or use the 3D View window as a basis for creating an animation of your simulation.

What would you like to do?

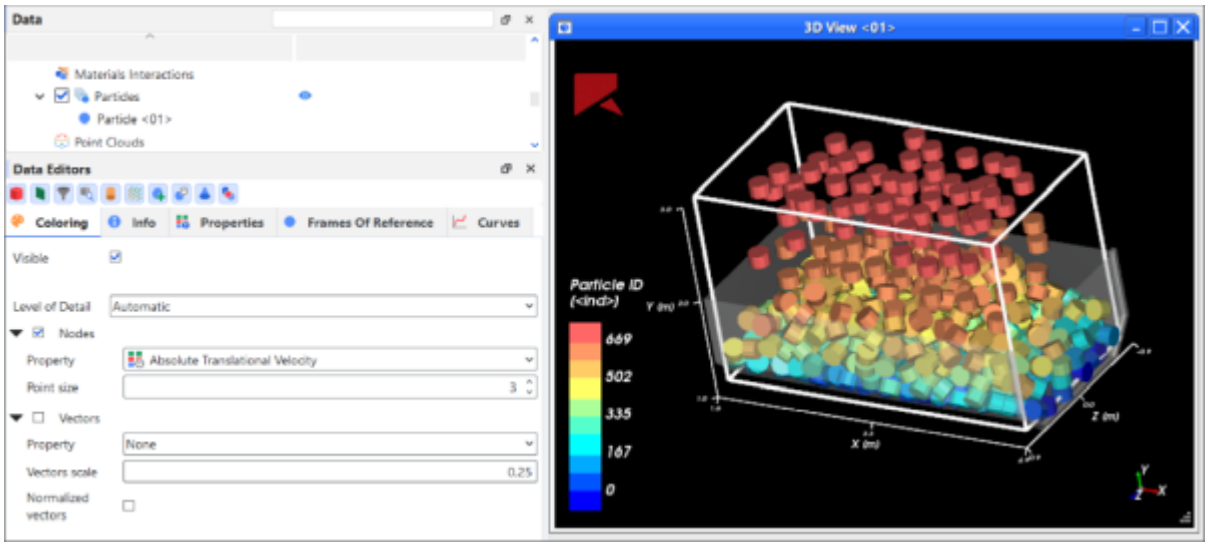
- Learn more About 3D View Windows (p. 1040)
- View Geometries, Particles, Points, and Fluids in 3D (p. 1041)
- Change a 3D View Window (p. 1042)
- Save a View with Custom Cameras (p. 1093)
- Reuse 3D View Window, Workspace, and User Process Setups in Other Projects (p. 403)
- Create and Save an Animation (p. 1166)

5.2.1. About 3D View Windows

A 3D View window is where in Rocky you will do most of your simulation visualization. Prior to processing, it is where you can preview your geometry, point cloud, and inlet placements. After processing, it is where you will see the particle flow and how it interacts with the geometries and other components in your design (as shown below).

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

Figure 5.45: 3D View window showing particle flow after processing



Tips:

- To preview your geometry motions prior to processing your simulation, use a Motion Preview window instead. (See also Preview a Motion in 3D (p. 573).)
- To preview a particle shape, or view on a representative particle shape collision statistics that you have collected during processing, use a Particles Details window instead. (See also Preview a Particle Shape in 3D (p. 681).)

As with other windows in Rocky, there are various ways you can change what appears in a 3D View window, including fonts, overlays, background colors, and grid lines. After you set up your 3D View window, you can choose to reuse the zoom, rotation, and pan settings in another window by saving and applying a Custom Camera Preset. You can also reuse your window settings within another project by exporting the project context. And, you can use a 3D View window as a basis for creating an animation that you can use to share your simulation results outside of Rocky.

What do you want to do?

- View Geometries, Particles, Points, and Fluids in 3D (p. 1041)
- Change a 3D View Window (p. 1042)
- Save a View with Custom Cameras (p. 1093)
- Reuse 3D View Window, Workspace, and User Process Setups in Other Projects (p. 403)
- Create and Save an Animation (p. 1166)

5.2.2. View Geometries, Particles, Points, and Fluids in 3D

1. Ensure that the components you want to show in the 3D View window have been added to the simulation. (See also Set Simulation Parameters (p. 414).)

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

2. From the **Window** menu, click **3D View**. A new 3D View window (p. 1396) appears in the Workspace (p. 1396).

Tip:

You can also create a new 3D View by clicking and dragging the component(s) you want to visualize from the **Data** panel to an empty location within the Workspace.

See Also:

- About 3D View Windows (p. 1040)
- Change a 3D View Window (p. 1042)
- Reuse 3D View Window, Workspace, and User Process Setups in Other Projects (p. 403)

5.2.3. Change a 3D View Window

There are several ways you can change what you see in the 3D View window once you have created it, as explained below.

- **Zoom, pan, tilt, and fit the image** by using your mouse, keyboard, an/or the Camera Visualization toolbar
- **Change the colors and data attributes of the image**, including the faces, edges, nodes, or vectors being displayed by using the Coloring tab on the Data Editors panel (or the Coloring toolbar)
- **Show/hide the simulation entity you want displayed** by using eye icons on the Data panel
- **Change the attributes of the 3D Window itself**, such as background colors, grid lines, and legend and caption details by using the Window Editors panel

What would you like to do?

- Learn more about Using the Mouse, Keyboard, or Toolbar to Change a the View (p. 1043)
- Learn more about Using the Window Editors Panel to Change a 3D View (p. 1047)
- Learn more about Using the Coloring Tab to Change a 3D View (p. 1074)
- Show/Hide Components by Using Eye icons and Checkboxes (p. 113)
- Learn more About Using the Coloring Toolbar to Change a 3D View (p. 1084)

See Also:

- View Geometries, Particles, Points, and Fluids in 3D (p. 1041)
- Save a View with Custom Cameras (p. 1093)
- Reuse 3D View Window, Workspace, and User Process Setups in Other Projects (p. 403)

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

Using the Mouse, Keyboard, or Toolbar to Change the View

Using the mouse and/or keyboard, or the Camera Visualization toolbar, there are several ways you can magnify (zoom), rotate (tilt), pan, and fit the data you see in the 3D View, Motion Preview, or Particles Details window window.

What would you like to do?

- Learn more About Using the Mouse and Keyboard to Change the View (p. 1043)
- Learn more About Using the Camera Visualization Toolbar (p. 1045)
- Use the Mouse, Keyboard, or Toolbar to Change a View (p. 1047)

ABOUT USING THE MOUSE AND KEYBOARD TO CHANGE THE VIEW

There are several ways you can use the mouse and keyboard to change what you see in the 3D View, Motion Preview, or Particles Details window. Note: Some—but not all—of these methods work for Plots as well.

Tip:

To see more keyboard shortcuts used in Rocky, refer to the Keyboard Accessibility and Shortcuts (p. 128) topic.

See the images and tables below for details.

Figure 5.46: Mouse with center scroll wheel



Table 1: Mouse and keyboard functions

Note:

The below interactions assume that the **Change vizu interactor configuration** button is set to **Default**. (See also About Using the Camera Visualization Toolbar (p. 1045).)

Function	Mouse Interactions
Zoom or magnify	To zoom in, position the mouse cursor over the view or plot and then move the scroll wheel (2) away from you. To zoom out, move the scroll wheel towards you.

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

Function	Mouse Interactions
Pan	To pan, position the mouse cursor over the view or plot, and then click and hold the right mouse button (3) or center wheel buttons (2) while moving the mouse.
Tilt or rotate	For views only, to tilt or rotate, position the mouse cursor over the image and then click and hold the left mouse button (1) while moving the mouse.
Change the location of the center point of rotation	For views only, to change the location of the point used for tilting or rotating, press the Ctrl key, and then with the left mouse button, click the new geometry location you want. The view will re-center to that new location and any rotations will happen around that new point.
Fit the view	To have Rocky automatically fit the data to the view or plot window, press the R key. Tip:Additional view-related keyboard combinations are listed on the **Shortcuts** section of the **Preferences** panel. (See also Set Global Preferences (p. 238).)
Inspect an individual *triangle* or *block*	For 3D Views of Geometries and Eulerian Statistics, you may double-left-click the triangle or block you want to inspect and a new Inspector User Process will be automatically created for the entity you chose. If there is already an Inspector User Process for that entity, Rocky will help you choose to update the cell location of the existing User Process, or create a new one. Note: Double-left-clicking a cell in a 3D View window does not currently work for Particles, 1-Way LBM, or 1-Way Fluent. (See also About the Cell Inspector (p. 1211).)
Move overlay or legend displays	For views only, position the mouse cursor over the middle of the overlay or legend you want to move, click and hold the left mouse button (1), drag the item to the new location, and then release the mouse. (See also About Using the Window Editors panel to Change a 3D View (p. 1058).)
Resize overlay or legend displays	For views only, position the mouse over the border of the overlay or legend you want to resize, click and hold the left mouse button (1), drag the border to the new location, and then release the mouse. (See also About Using the Window Editors panel to Change a 3D View (p. 1058).)

What would you like to do?

- Use the Mouse, Keyboard, or Toolbar to Change a View (p. 1047)
- Use the Mouse, Keyboard, or Toolbar to Change a Plot (p. 1125)
- Learn more About Using the Camera Visualization Toolbar (p. 1045)

See Also:

- Change a 3D View Window (p. 1042)
- Change a Particles Details Window (p. 682)
- Change a Motion Preview Window (p. 575)
- Changing the Appearance of a Graph (Plot or Histogram) (p. 1125)

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

ABOUT USING THE CAMERA VISUALIZATION TOOLBAR

The Camera visualization toolbar (shown below) has several buttons that are useful for visualizing the objects in your 3D View, Motion Preview, or Particles Details windows—and even some Plot windows—including camera preset views, spin, and projection abilities.

Use the images and table below to help you understand how to use this toolbar for adjusting what you see in your views.

Figure 5.47: Camera Visualization toolbar



Table 1: Camera Visualization toolbar button descriptions

Button	Description
 Fit	For all windows, clicking the **Fit** button automatically adjusts the zoom and pan of the selected view or plot window to show any data or entities (p. 1395) that are marked as visible. (See also Show/Hide Components by Using Eye icons and Checkboxes (p. 113)). For view windows only, clicking the down arrow to the right of the **Fit** button displays several Camera Presets that you can use to quickly snap-to a particular view. Specifically: • **+ X**: Looking down the X axis from the positive end. • **+ Y**: Looking down the Y axis from the positive end. • **+ Z**: Looking down the Z axis from the positive end. • **- X**: Looking down the X axis from the negative end. • **- Y**: Looking down the Y axis from the negative end. • **- Z**: Looking down the Z axis from the negative end. • **+ X 30**: Looking down the X axis from the positive end with a negative 30 degree rotation on the Y axis. • **+ Y 30**: Looking down the Y axis from the positive end with a negative 30 degree rotation on the X axis.
 Undo camera modification Redo camera modification	For 3D View windows only, these buttons enable you to undo or redo any camera-related modifications you made to any open 3D View window. The changes include ones made with the mouse and keyboard (see also About Using the Mouse and Keyboard to Change the View (p. 1043)), or through the Camera Presets options on the **Fit** button menu.
 Camera spin	For view windows only, when the **Camera spin** button is enabled (pressed in), it allows all views to spin in the speed and direction you indicate by

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

Button	Description
	left-clicking and dragging (slowly or quickly) then releasing the mouse on the view. Tip:Left-click the view again to stop the spinning. When the **Camera spin** button is disabled, left-clicking and dragging the mouse rotates the view but will not spin it upon release of the mouse.
 Change projection: Default	For view windows only, when the **Change projection:Default/Orthogonal** button is set to Default, a perspective projection is provided of all open views, showing distant objects foreshortened to provide additional realism.
 Change projection: Orthogonal	For view windows only, when the **Change projection:Default/Orthogonal** button is set to Orthogonal, a parallel projection is provided of all open views, showing to-scale objects that ignore perspective.
 Change vizu interactor configuration	For view windows only, changes how the mouse buttons and center wheel affects items being shown in all open view windows. Specifically, when set to: • **Default**, the view responds as indicated to the following mouse and keyboard combinations: – Left mouse button: Rotates the view – Right mouse button: Pans the view – Center wheel: Zooms the view – Left + right mouse buttons: Zoom the view – Ctrl + left mouse button: Changes the location of the view's center point – Shift + left mouse button: Displays cell and geometry component ID information (3D Views only) • **No Rotate**, the view responds as indicated to the following mouse and keyboard combinations: – Left mouse button: Pans the view – Right mouse button: Zooms the view – Center wheel: Zooms the view – Left + right mouse buttons: Zoom the view – Ctrl + left mouse button: Changes the location of the view's center point – Shift + left mouse button: Displays cell and geometry component ID information (3D Views only) (See also About Using the Mouse and Keyboard to Change the View (p. 1043)).

What would you like to do?

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

- Use the Mouse, Keyboard, or Toolbar to Change a View (p. 1047)
- Use the Mouse, Keyboard, or Toolbar to Change a Plot (p. 1125)
- Learn more About Using the Mouse and Keyboard to Change the View (p. 1043)

See Also:

- Change a 3D View Window (p. 1042)
- Change a Particles Details Window (p. 682)
- Change a Motion Preview Window (p. 575)
- Changing the Appearance of a Graph (Plot or Histogram) (p. 1125)

Use the Mouse, Keyboard, or Toolbar to Change a View

1. Select the 3D View, Motion Preview, or Particles Details window (p. 1396) you want to change.
2. Do one or more of the following:
   - Use the left, right, or center wheel buttons on the mouse to rotate, pan, or zoom respectively.
   - Ctrl + left-click a geometry or particle shape location to change the center point of rotation.
   - Press R to automatically fit the window view to the available data.
   - Change to another camera preset by pressing x, X, y, Y, z, and Z respectively. Note: This is the same as using the options on the **Fit** menu, which is located on the Camera Visualization toolbar. (See also About Using the Camera Visualization Toolbar (p. 1045).)
   - Double-left-click a geometry triangle (p. 1395) or Eulerian Statistics block (p. 1389) to inspect that individual cell.

Tip:

Additional keyboard shortcuts are listed in the Preferences dialog. (See also Keyboard Accessibility and Shortcuts (p. 128)).

See Also:

- About Using the Mouse and Keyboard to Change the View (p. 1043)
- Use the Mouse, Keyboard, or Toolbar to Change a Plot (p. 1125)

Using the Window Editors Panel to Change a 3D View

From the Window Editors panel, you can modify display settings for all the 3D View windows in your project, or just the selected 3D View window. Changes can include the background color, font color/sizes, text and/or image overlays, and window axes displays. You can also synchronize the time steps displayed when two or more 3D View windows are open.

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

What would you like to do?

- Learn more About Using the Window Editors Panel to Change All 3D Views in the Project (p. 1048)
- Change Window Display Options for All 3D Views in the Project (p. 1058)
- Learn more About Using the Window Editors Panel to Change the Selected 3D View (p. 1058)
- Change Window Display and Data Update Options for the Selected 3D View (p. 1068)
- Add or Edit a Text Overlay for a 3D View Window (p. 1069)
- Add or Edit an Image Overlay for a 3D View Window (p. 1070)

See Also:

- Change a 3D View Window (p. 1042)

ABOUT USING THE WINDOW EDITORS PANEL TO CHANGE ALL 3D VIEWS IN THE PROJECT

From the Window Editors panel, you can modify display settings for all the 3D View windows in your project. These include settings such as Property legend display options, window axes display options, and Particle rendering scale.

Tip:

You can also use Rocky Preferences to change the background color and fonts for all the 3D View windows in your project. (See the "Preferences for Windows Properties" section in the About Setting Global Preferences (p. 227) topic.)

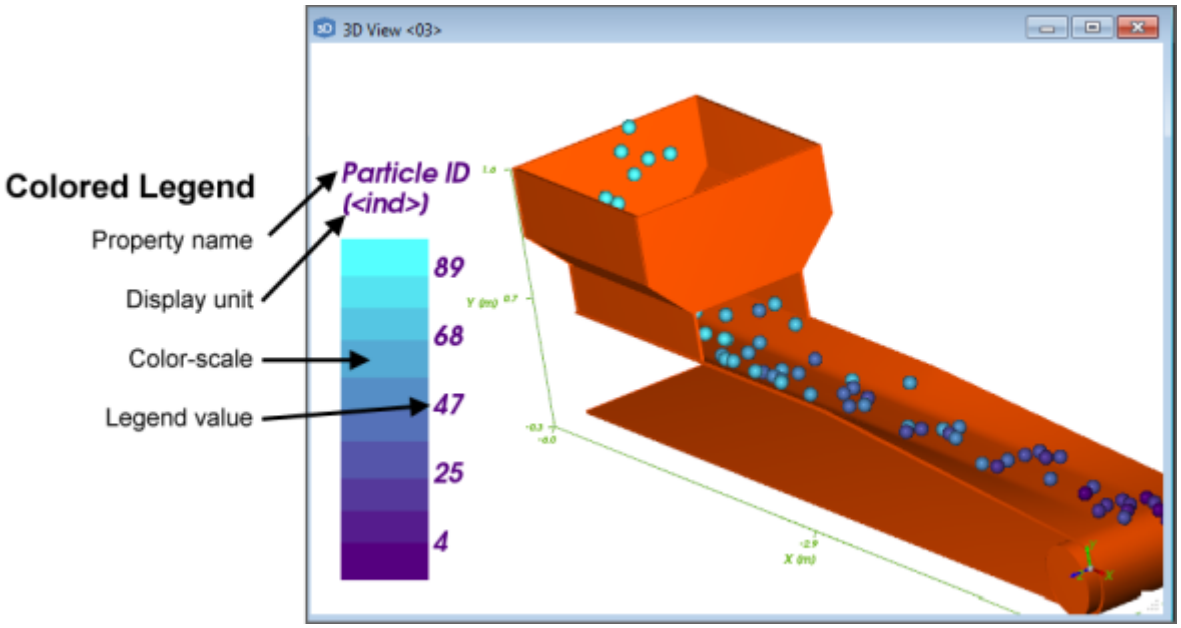
You can also duplicate the 3D window to facilitate the viewing and editing of different parts of your project. To do this, simply right-click on the tab of the 3D window you want to duplicate and click in **Duplicate**.

PROPERTIES DISPLAY OPTIONS

The options on the Properties tab enable you to set the colored legend display options for each of the various Properties that are listed. The image below illustrates the options that can be changed.

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

Figure 5.48: Colored Legend options that can be changed



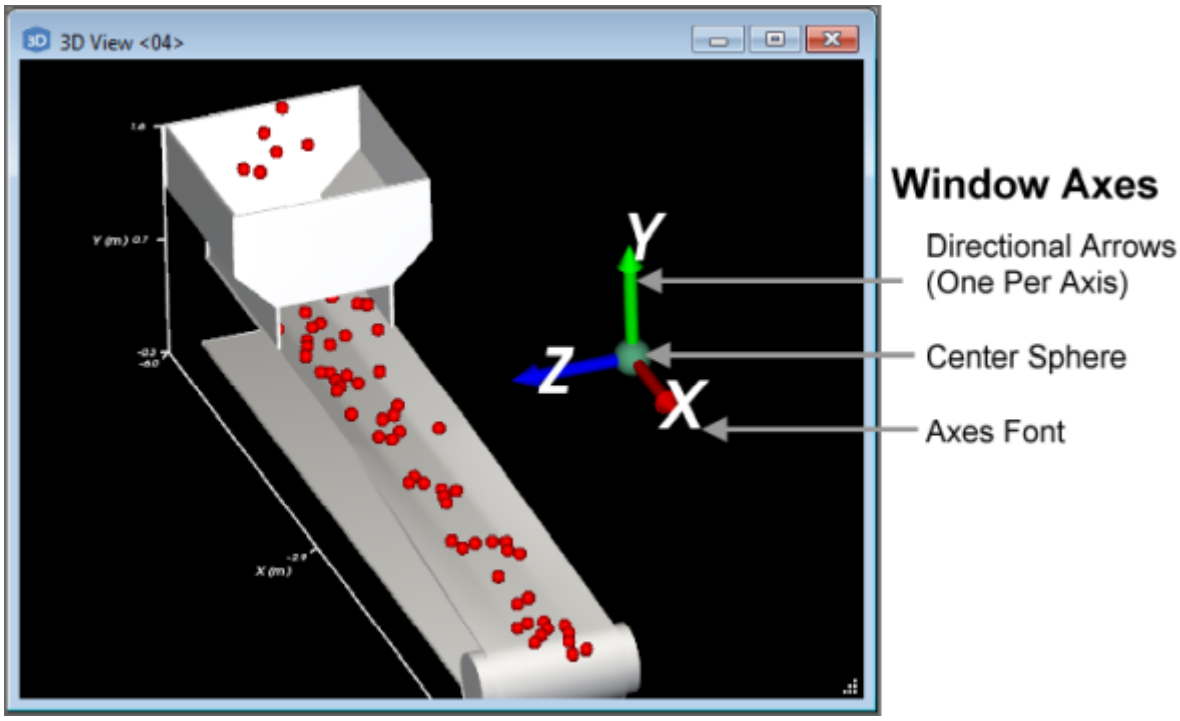
These include the colors making up the banding, the number of value ticks, and the legend units displayed—all of which apply to all 3D Views in your project, and will also be replicated on the Colorings tab of the associated Color Scale. (See also About Color Scales (p. 1086).)

AXES DISPLAY OPTIONS

The options on the Axes tab enable you to change the display of the set of colored axes on each of your 3D View windows. The settings include changing individual axis arrow colors, and making changes to the axes set as a whole, including font colors and sizes, as well as the size and placement of the axes set itself:

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

Figure 5.49: Window Axes options that can be changed



Important The settings you make on the Axes tab for any of the three window types in Rocky—including the 3D View, Motion Preview, and Particles Details windows—will apply to the other two window types respectively. In addition, any changes you make to an Axes tab will also be replicated in future Rocky projects. (See also About Using the Window Editors Panel to Change the Window Axes Displays (p. 1071).)

PARTICLES DISPLAY OPTIONS

The **Rendering Scale** option on the Particles tab enables you to make particles appear slightly larger or smaller in 3D View windows than calculated.

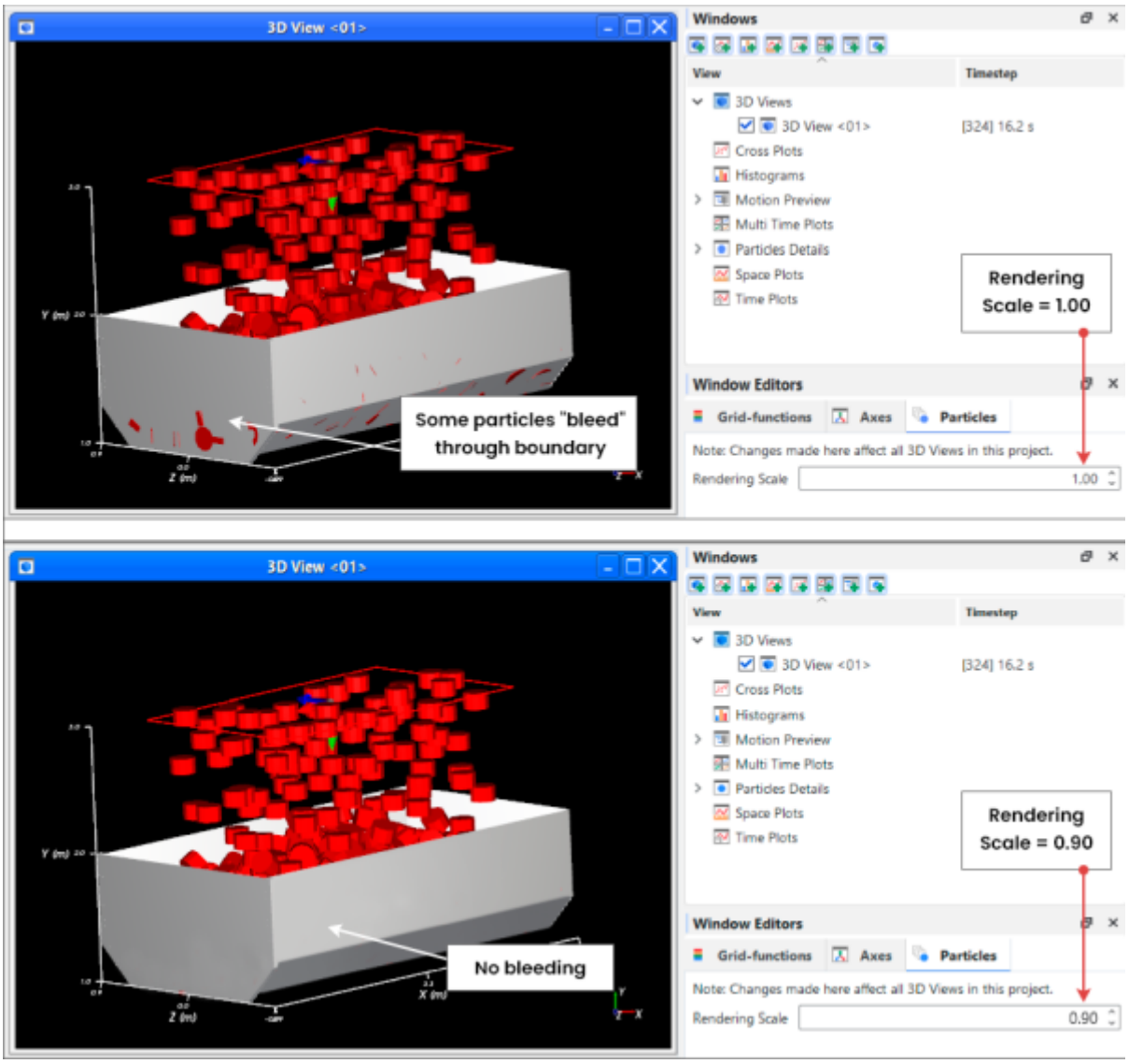
Important:

This rendering affects only the visualization of the particles and has no effect upon the results nor the analyses.

Increasing the scale can be used in cases where the particles in the simulation are so small that you want to exaggerate the size to better show the shape. Decreasing the scale can be a useful in cases where particles appear to "bleed" through the boundary, as illustrated in below.

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

Figure 5.50: Particle rendering scale options



This kind of visual overlap between the particles and the geometry can occur, for example, when the geometries are imported as a surface. Because a surface boundary has no thickness, the particles can sometimes appear to come through it.

In the example shown above, the Rendering Scale is changed from the default of 1.00 to the new value of 0.90, which reduces the rendering of the particles by 10%. In this case, this is enough of a reduction to prevent the Particles from appearing to go through the boundary.

PROJECT-WIDE 3D VIEW SETTINGS ON THE WINDOW EDITORS PANEL

See the images and table below to understand more about how you can use the Window Editors panel to change all 3D View windows in your project.

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

Figure 5.51: Grid-functions options on the Window Editors panel when the main 3D View entity is selected

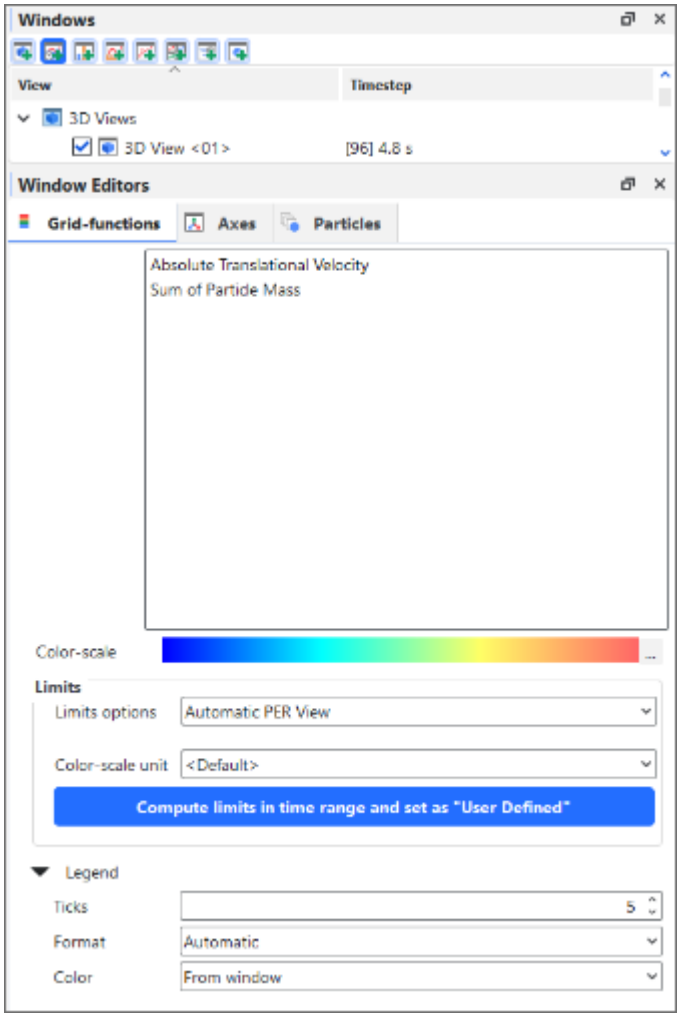


Table 1: Properties options on the Window Editors panel

Setting	Description	Range
Properties	List of Rocky Properties that can be displayed in 3D Views.	Automatically provided
Color-scale	Displays the current color banding configuration for the legend associated with the selected Property. Clicking the **...** button will enable you to edit the options. (See Figure 5.52: Color-scale dialog (p. 1054) and Table 2 below.)	Display only
Limits		
Limits options	Determines how the minimum and maximum values for the legend associated with the selected Property are assigned. Specifically: • **Automatic PER view** enables Rocky to determine, for the selected 3D View only, the limit values automatically based upon the current time step.	Automatic PER view; Automatic ALL views; User Defined

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

Setting	Description	Range
	• **Automatic ALL views** enables Rocky to determine, for all 3D Views using the selected Color Scale, the limit values automatically based upon the current time step. • **User Defined** enables you to manually set the limit values. **Tip:** You may also have Rocky help you calculate the limits by specifying a time range based upon an applicable process(es). To do so, click the **Compute limits in time range and set as "User Defined"** button. (See also Figure 5.53: Calculate limits for time range dialog (p. 1055) and Table 3 below.)	
Limits	When **Limit options** is set to **User Defined**, this enables you to enter the minimum and maximum values you want to define the legend's limits.	Any value
Color-scale unit	Determines the units that will be displayed in the legend. Note: If the **Color-scale unit** differs from the unit you selected in the **Limits** drop-down list, your limit values will be translated into this preferred unit on the actual legend display.	Options limited by list box
Legend		
Ticks	Defines the number of divisions whose respective values will be displayed next to the legend. **Note:** Ticks represent only the numerical divisions of the legend. To change the color divisions, use the Banded option on the Color-scale dialog. (See Table 2 below.)	1 > 65
Format	Enables you to change how the values next to each tick are formatted. Specifically: • **Integer** displays full values without decimals. • **Decimal** options display the specified number of digits following the decimal. • **Exponential** options display the specified number of digits prior to the exponent. • **Automatic** displays the Rocky default format for the values.	Integer; Decimal 2 digits; Decimal 4 digits; Decimal 8 digits; Exponential 2 digits; Exponential 4 digits; Exponential 8 digits; Automatic

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

Setting	Description	Range
Color	Defines how font color of the values listed next to the legend is determined. Specifically: • **From window** uses the same font color as defined for the 3D View window itself. • **Custom** enables you to select a different color for only the values displayed next to the legend.	From window; Custom

Figure 5.52: Color-scale dialog

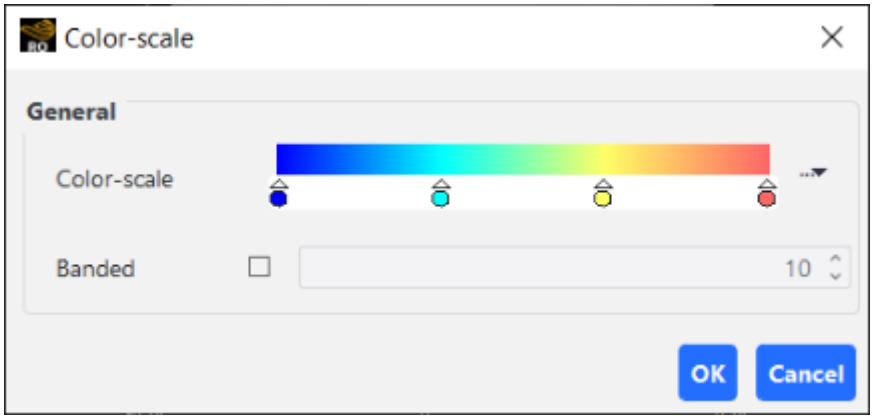


Table 2: Color-scale dialog options

Setting	Description	Range
Color-scale	Determines the gradient distribution of color along the selected scale. From here, you can move the color sliders to adjust the gradient, double-click color slider handles to change the color, double-click the slider bar itself to add another color slider, or click the **...** button to select from several additional presets, which you can also modify.	Limited by the UI
Banded	When selected, the legend will divide the color scale into the number of solid bands you indicate. When cleared, the legend will display the color scale in a smooth, unbanded gradient. **Note:** Banded applies only to the color divisions of a legend. To change the numerical divisions, change the Ticks value instead. (See Table 1 above.)	Turns on or off; Positive whole numbers 2 or greater

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

Figure 5.53: Calculate limits for time range dialog

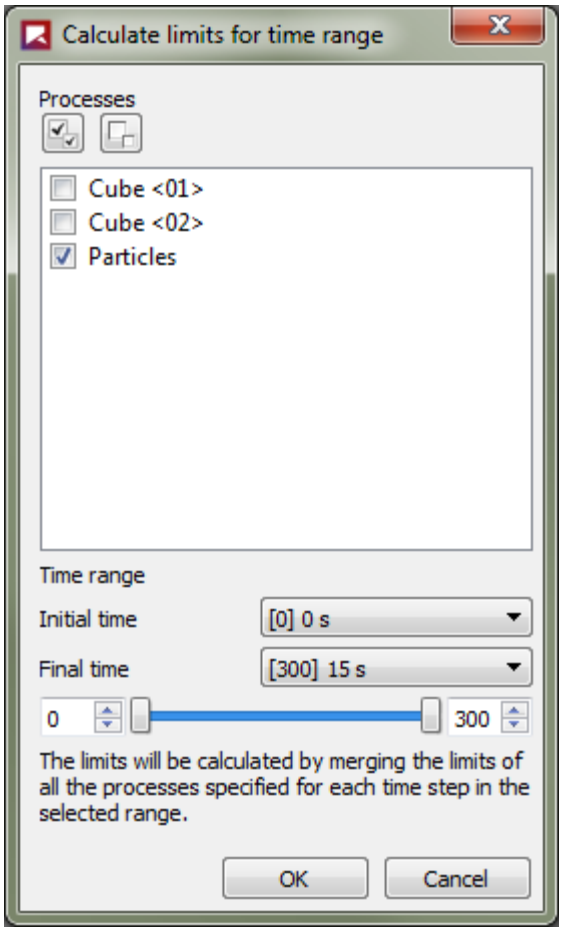


Table 3: Calculate limits for time range dialog options

Setting	Description	Range
Processes	Lists the applicable entities or User Processes upon which you can base your time range. You must select at least one process to calculate limits using a time range.	Limited by the UI
Time range		
Initial time	The time step with which to begin the time range. This can be set by selecting the time step you want from the drop-down list, by entering the corresponding output number for the time step, or by using the slider.	Limited by the UI
Final time	The time step with which to end the time range. This can be set by selecting the time step you want from the drop-down list, by entering the corresponding output number for the time step, or by using the slider.	Limited by the UI

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

Figure 5.54: Axes options on the Window Editors panel when the main 3D View entity is selected

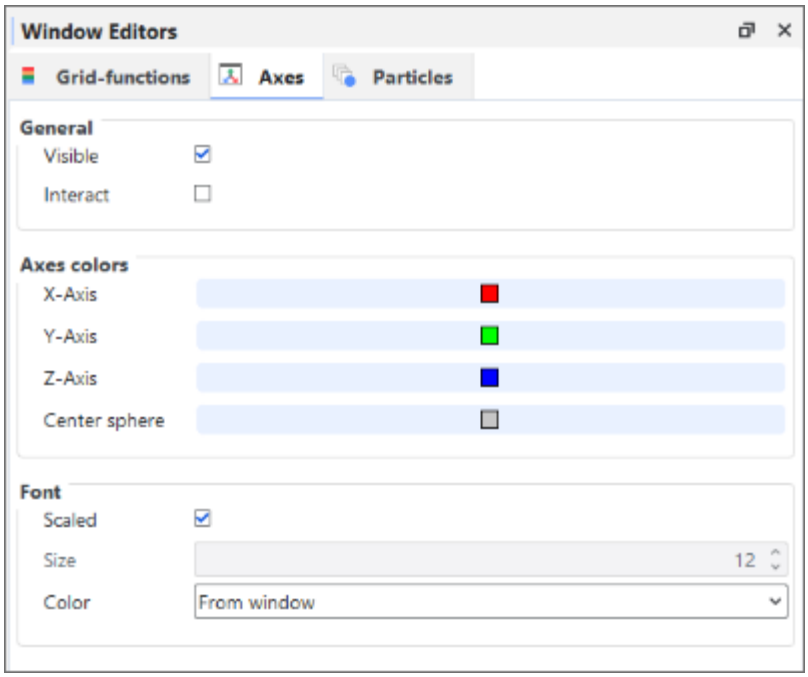


Table 4: Axes options on the Window Editors panel

Setting	Description	Range
General		
Visible	When enabled, shows a set of colored axes on all 3D View windows in your project.	Turns on or off
Interact	When enabled (and the **Visible** option is enabled as well), this allows you to move and change the size of the set of colored axes on all 3D View windows in your project.	Turns on or off
Axes colors		
X-Axis	Enables you to set the color of the axis arrow that represents the X coordinate direction.	Limited by the options available in the **Select Color** dialog.
Y-Axis	Enables you to set the color of the axis arrow that represents the Y coordinate direction.	Limited by the options available in the **Select Color** dialog.
Z-Axis	Enables you to set the color of the axis arrow that represents the Z coordinate direction.	Limited by the options available in the **Select Color** dialog.
Center sphere	Enables you to set the color of the sphere at the center of the axes.	Limited by the options available in the **Select Color** dialog.
Font		

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

Setting	Description	Range
Scaled	When enabled, the size of the text used on the axes set will be determined in relation to the height and width of the 3D View window itself; the axes text will grow or shrink in scale with the size of the window. When cleared, the font size can be set through the **Size** text box.	Turns on or off
Size	The value entered here determines the size of the axes text in relation to the computer screen. With this option, the text (and its sizing box) stays the same size no matter the height and width of the 3D View window that contains it.	Whole positive values
Color	Defines how the font color of the axes text is determined. Specifically: • **From window** uses the same font color as defined for the 3D View window itself. • **Custom** enables you to select a different color for only the text used with the axes.	From window; Custom

Figure 5.55: Particles options on the Window Editors panel when the main 3D View entity is selected

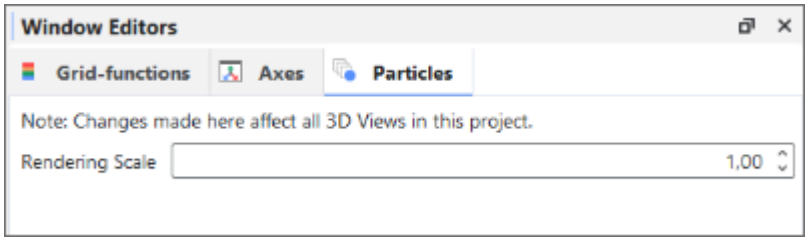


Table 5: Particles options on the Window Editors panel

Setting	Description	Range
Rendering Scale	Defines how Particles shown in 3D View windows will be rendered. Specifically: • The default value of 1.00 will render the Particles at 100% the defined size. • Values less than 1.00 will render the Particles at a smaller size than defined. For example, setting 0.97 will reduce the display of each Particle by 3%.	Values 0.01 or greater. **Tip:** Values closer to 1.00

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

Setting	Description	Range
	• Values greater than 1.00 will render the Particles at a larger size than defined. For example, setting 1.03 will increase the display of each Particle by 3%. **Important:** This setting affects the visualization of the Particles only and has no effect on the underlying calculations nor simulation results. **Note:** This text field does not support parametric variables. (See also Change Window Display Options for All 3D Views in the Project (p. 1334).)	are recommended.

What would you like to do?

- Change Window Display Options for All 3D Views in the Project (p. 1058)
- Learn more About Using the Window Editors Panel to Change the Selected 3D View (p. 1058)

See Also:

- Using the Window Editors Panel to Change a 3D View (p. 1047)
- About Color Scales (p. 1086)

CHANGE WINDOW DISPLAY OPTIONS FOR ALL 3D VIEWS IN THE PROJECT

1. Ensure that both the **Windows** and **Window Editors** panels are visible. (From the **View** menu, click **Windows**. From the **View** menu, click **Window Editors**.)
2. From the **Windows** panel, select the top most **3D Views** entity.
3. From the **Window Editors** panel, change the options you want.

See Also:

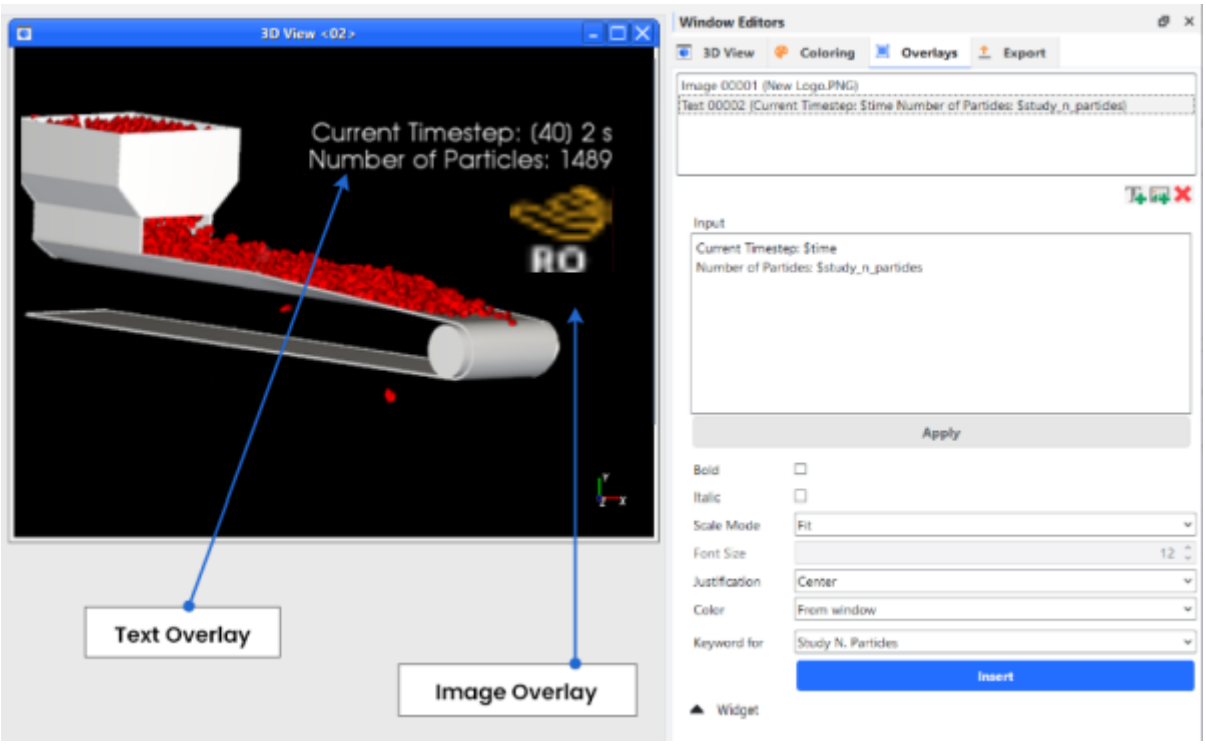
- About Using the Window Editors Panel to Change All 3D Views in the Project (p. 1048)
- About Using the Window Editors Panel to Change the Selected 3D View (p. 1058)
- Add or Edit a Text Overlay for a 3D View or Particles Details Window (p. 1069)
- Add or Edit an Image Overlay for a 3D View or Particles Details Window (p. 1070)

ABOUT USING THE WINDOW EDITORS PANEL TO CHANGE THE SELECTED 3D VIEW

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

When you use the Window Editors panel to change the selected 3D View, you have the ability to change how items in the window appear on the screen. These items include background color, font color/sizes, and text and/or image overlays (as shown below). You can also change how multiple windows of the same type respond to changes in time steps.

Figure 5.56: Examples of text and image overlays in a 3D View window



Tips:

- You can also use Rocky Preferences to change the background color and fonts for all the 3D View windows in your project, not just the selected one. (See the "Preferences for Windows Properties" section in the About Setting Global Preferences (p. 227) topic.)

- You can also change the Property legend display options, window axes display options, and Particle rendering scale for all the 3D View windows in your project. (See also About Using the Window Editors Panel to Change All 3D Views in the Project (p. 1048).)

In this section, we will cover only the items on the **3D View** tab and the **Overlay** tab. See below for other resources:

- The **Coloring** tab in the Window Editors panel for 3D View windows is almost identical to the one provided for default conveyors or imported geometry components on the Coloring tab of the Data Editors panel. See About Using the Coloring Tab to Change a View (p. 1074) to understand more about that tab.

- The **Export** tab will be covered in the About Exporting Data and Images Out of Rocky (p. 1155) topic.

You use the **3D View** tab on the **Window Editors** panel to change how the window appears and how the data displayed within it is updated. These options include changing display items like the background color; labels, lines, and axes colors; color scale label sizes; and window height and width. In addition, you can choose to show or hide the **Bounding box**, which displays measurements illus-

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

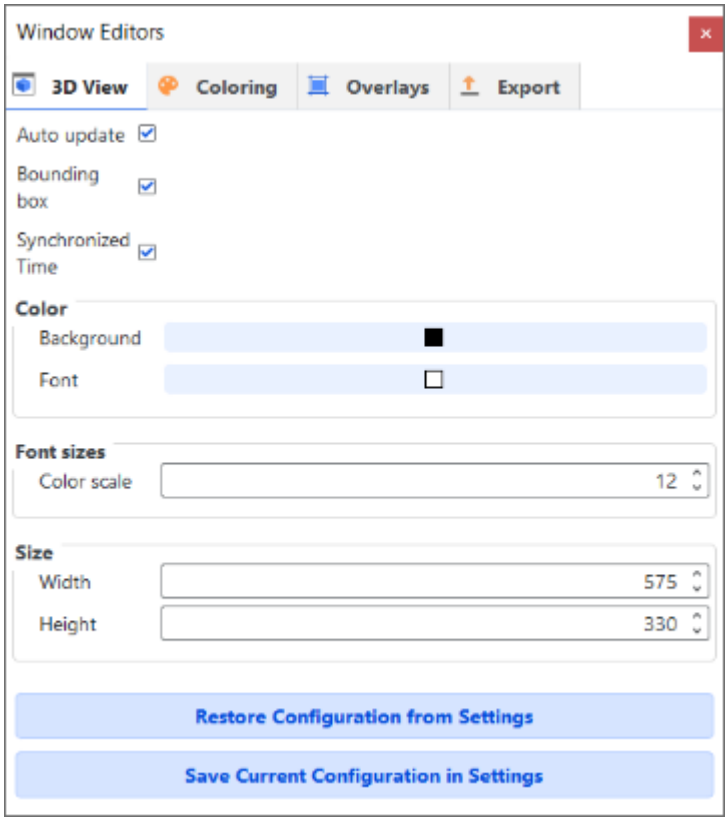
trating the simulation boundaries; use **Auto update** to enable or prevent the view from being updated when data affecting it is changed; or choose to have multiple views updated with the same time step by using **Synchronized Time**.

You use the **Overlays** tab on the **Window Editors** panel to add text or image overlays. Text overlays can include your own custom text or can include Keywords that will show data, including Property information and Input/Output variables, directly on your window view. Image overlays can include company or association logos, photos of real-world equipment, or any applicable image that compliments your simulation. Adding these kinds of overlays are especially useful for generating videos and images from your simulation.

3D VIEW AND OVERLAYS TAB SETTING DEFINITIONS

See the images and table below to understand more about how you can use the Window Editors panel to change a 3D View window.

Figure 5.57: 3D View options on the Window Editors panel



Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

Figure 5.58: Initial Overlay options for 3D View windows on the Window Editors panel

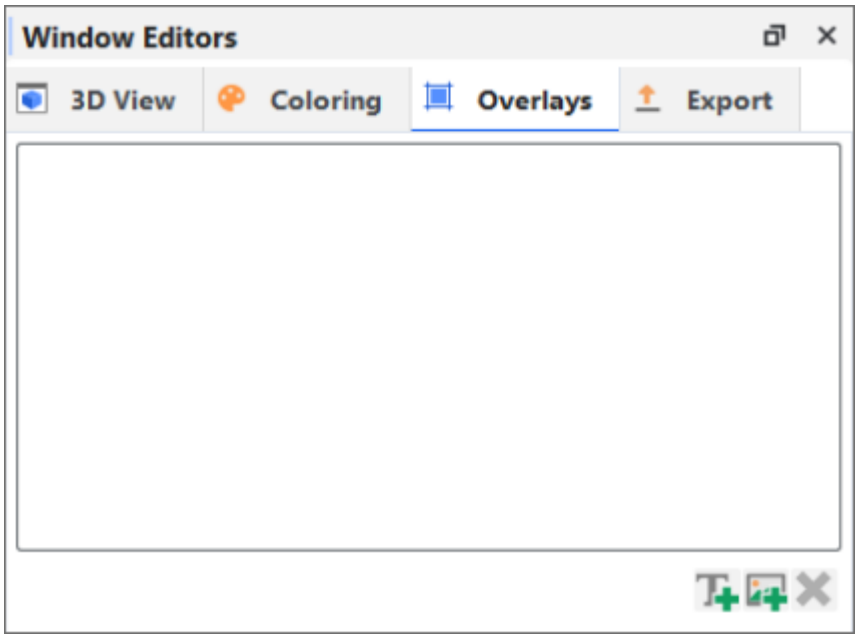
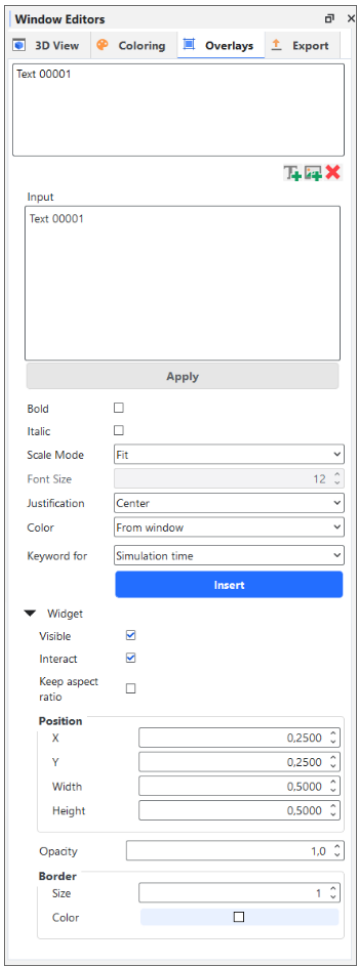


Figure 5.59: Text Overlay options on the Window Editors panel



Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

Figure 5.60: Image Overlay options on the Window Editors panel

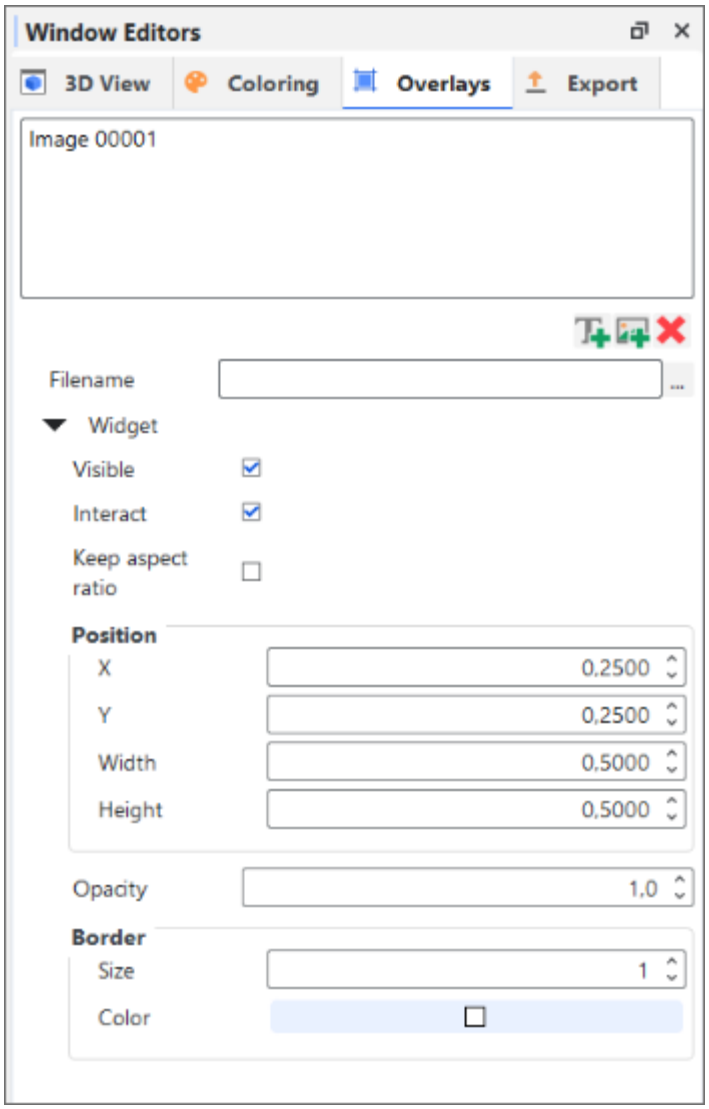


Table 1: 3D View tab options on the Window Editors panel

Setting	Description	Range
Auto update	When enabled, any change made to the display settings of the selected window or to the items affecting the data or settings calculated and displayed within the window, including the time step or Properties displayed, will be updated in the window automatically. Because automatically updating the view with each separate change can be computationally intensive, you can clear this option to have all calculated items and more computationally intensive display options remain	Turns on or off

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

Setting	Description	Range
	unchanged in the window until you enable **Auto update** again. **Tip:** You will know when the calculations in a window are not being updated when you see a thick red border around the window.	
Bounding box	When enabled, displays measurements illustrating the simulation boundaries in the selected window.	Turns on or off
Synchronized Time	When enabled and the window (or any other window with this checkbox enabled) is selected, the details shown in this (and any other Synchronized Time) window will be updated when the current time step is changed on the Time toolbar. (See also Use The Time Step Toolbar (p. 1280)).When cleared and the window is selected, only this window will be updated when the current time step is changed on the Time toolbar. **Tip:** To keep the time step synchronized between multiple windows, ensure each window has this option enabled.	Turns on or off
Color Background	Enables you to change the color that appears behind the geometries and particles in the window.	Options limited by the choices in the **Select Color** dialog
Color Font	Enables you to change the color of the labels, borders, and axes lines displayed in the window. **Note:** These color options can be overridden for text overlays by using the **Color** option on the **Overlays** tab. (See Table 2 below.)	Options limited by the choices in the **Select Color** dialog
Font sizes Color scale	Enables you to change the size of the text used on color scales for Properties, which, unlike the other lines or labels displayed on the window, is not affected by the window size or the level of zoom you have applied.	Whole values from 1 to 100
Size Width	Enables you to numerically change how wide the window is. Useful in cases where you need exact dimensions for generating animations, or ensuring image consistency.	Whole positive values between 123 and 5000.

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

Setting	Description	Range
Size Height	Enables you to numerically change how tall the window is. Useful in cases where you need exact dimensions for generating animations, or ensuring image consistency.	Whole positive values between 0 (zero) and 5000.
Restore Configuration from Settings	Clicking this button replaces the values set on the tab with the ones that have been saved to the (internal) Rocky Settings folder. (See also Rocky File Types and Folders (p. 129)).	(Button selection)
Save Current Configuration in Settings	Clicking this button overwrites the values that have been saved to the (internal) Rocky Settings folder with the ones currently set on the tab. (See also Rocky File Types and Folders (p. 129)).	(Button selection)

Table 2: Overlay tab options on the Window Editors panel for Text and Image Overlays

Setting	Description	Range
Text Overlays Only		
Input	This box displays the text, phrase, or keyword that you want overlaid onto the selected window. It can be static text that you type in, can be dynamic text by using the **Keyword for** list, or can be a combination of both.	No limit
Bold	Enables the selected text overlay to appear bold in the window.	Turns on or off
Italic	Enables the selected text overlay to appear italic in the window.	Turns on or off
Scale Mode	Determines how the text in the selected overlay scales in relation to its **Font Size**, the sizing box that surrounds it, and size of the window itself. Specifically: • **Font Size** uses the value in the **Font Size** box to set the size of the text in relation to the computer screen. With this option, the text (and its sizing box) stays the same size no matter the height and width of the window that contains it. • **Fit** adjusts the size of the text based upon the sizing box that surrounds it. With this option, the text grows or shrinks in scale with the height and width of its sizing box. • **Scaled Font Size** uses the value in the **Font Size** box to set the size of the text in relation to the height and width of the window. With this option, the text grows or shrinks in scale with the size of the window.	Font Size; Fit; Scaled Font Size
Font Size	When **Font Size** or **Scaled Font Size** are chosen for **Scale Mode**, this sets the size of the text in the selected overlay.	Whole positive values

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

Setting	Description	Range
Justification	Aligns the selected text horizontally in the overlay box. **Tip:** This feature is really only useful when you several lines of text. Use hard returns (press the Enter key) to force text to the next line.	Left Center Right
Color	Enables you to indicate from what location the font color overlay text will be taken. Specifically: • **From window** uses the **Color Font** settings in the **(Window Type)** tab. (See Table 1 above.) • **Custom** enables you to choose a unique color separate from the other text used in the window.	From window Custom
Keyword for	Enables you to enter Rocky-specific placeholder values for data and settings that pertain to your simulation. As your simulation processes or as you change your time step, and assuming you have selected **Auto update** on the **(Window Type)** tab (see Table 1 above), these values will be updated automatically in the text overlay. In addition to the items explained below, if you have defined any Input or Output Variables (see also About Defining and Using Input Variables (p. 243) and About Defining Output Variables (p. 250)), these will be listed here as well. See below for more information on keywords. For more information on Properties, see About Properties (p. 933) . • **Property maximum value (Geometries)** lists the highest value calculated thus far for the last Property pertaining to a default (feed or receiving conveyor) or imported geometry you have applied to the selected window. • **Property maximum value (Particles)** lists the highest value calculated thus far for the last Property pertaining to Particles that you have applied to the selected window. • **Property minimum value (Geometries)** lists the lowest value calculated thus far for the last Property pertaining to a default (feed or receiving conveyor) or imported geometry you have applied to the selected window. • **Property minimum value (Particles)** lists the lowest value calculated thus far for the last Property Function pertaining to Particles that you have applied to the selected window. • **Property name (Geometries)** lists the name, for example, Instantaneous Impact Power, of the last Property pertaining	Property maximum value (Geometries); Property maximum value (Particles); Property minimum value (Geometries); Property minimum value (Particles) Property name (Geometries); Property name (Geometries); Property name (Particles); Property unit (Geometries); Property unit (Particles); Simulation time; Simulation time index; Simulation time value;

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

Setting	Description	Range
	to a default (feed or receiving conveyor) or imported geometry you have applied to the selected window. • **Property name (Particles)** lists the name, for example, Absolute Rotational Velocity, of the last Property pertaining to Particles that you have applied to the selected window. • **Property units (Geometries)** lists the units used by the last Property pertaining to a default (feed or receiving conveyor) or imported geometry you have applied to the selected window. • **Property units (Particles)** lists the units used by the last Property pertaining to Particles that you have applied to the selected window. • **Simulation time** lists the current time step information presented in the same format as provided on the Time toolbar. This includes the number of *output files* saved, the current output time value, and the time step units. For example, **(5) 0.25 s** represents the point in the simulation where 5 output files were saved after 0.25 seconds. (See also About the Time Toolbar (p. 1279)). • **Simulation time index** lists the number of time step output files that were saved at the current time step selected. • **Simulation time value** lists the particular moment of Simulation time at the current time step selected. For example, **0.25 s**. • **Study Customer Name** lists what you have entered in the **Customer Name** box on the **Study** portion of the **Data** panel. (See also Study Parameters (p. 415)). • **Study Description** lists what you have entered in the **Description** box on the **Study** portion of the **Data** panel. (See also Study Parameters (p. 415)). • **Study N. Particles** lists the total number of whole particles that are located within the simulation boundaries at the current time step. • **Study Name** lists what you have entered in the **Study Name** box on the **Study** portion of the **Data** panel. (See also Study Parameters (p. 415)). **Tip:** If you find that it takes too long to update the text displayed when changing time steps or other data,	Study Customer Name; Study Description; Study N. Particles; Study Name

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

Setting	Description	Range
	consider clearing the **Auto update** checkbox on the **(Window Type)** tab until your changes are complete. (See Table 1 above.)	
Image Overlays Only		
Filename	This is the location and file name of the image you want overlaid onto the selected window.	.bmp .jpg .png .pnp
Widget (both Text and Image Overlays)		
Visible	When enabled, shows the selected overlay in the window. When cleared, it hides the overlay.	Turns on or off
Interact	When enabled, allows you to move and resize the selected overlay. When cleared, it locks the size and position.	Turns on or off
Keep aspect ratio	When enabled, locks the aspect ratio of the sizing box surrounding the overlay so that when you resize it, the shape of the box is maintained. When cleared, you may change the shape of the sizing box surrounding the overlay. **Note:** The shape of the sizing box has no real effect upon the display of the overlay. No matter the shape, Rocky will automatically maximize the size of the content to fill it.	Turns on or off
Position | X	For the selected overlay, this defines the X-position of the sizing box's lower left corner normalized in relation to the window size. A value of zero (0) is equal to the left window border and a value of 1 is equal to the right window border.	0-1
Position | Y	For the selected overlay, this defines the Y-position of the sizing box's lower left corner normalized in relation to the window size. A value of zero (0) is equal to the lower window border and a value of 1 is equal to the upper window border.	0-1
Position | Width	For the selected overlay, this defines the width of the sizing box normalized in relation to the window size. A value of 1 is equal to the full width of the window.	0-1
Position | Height	For the selected overlay, this defines the height of the sizing box normalized in relation to the window size. A value of 1 is equal to the full height of the window.	0-1

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

Setting	Description	Range
Opacity	Enables you to adjust the transparency of the overlay. The higher the value, the more opaque the overlay appears. The lower the value, the more transparent.	0.0 - 1.0 (Values limited by UI)
Border Size	Enables you to increase the thickness of the sizing box that surrounds the overlay.	Whole positive values including 0 (zero)
Border Color	Enables you to change the color of the sizing box that surrounds the overlay.	Options limited by the choices in the **Select Color** dialog.

What would you like to do?

- Change Window Display and Data Update Options for the Selected 3D View (p. 1068)
- Add or Edit a Text Overlay for a 3D View or Particles Details Window (p. 1069)
- Add or Edit an Image Overlay for a 3D View or Particles Details Window (p. 1070)
- Learn more About Using the Coloring Tab to Change a View (p. 1074)
- Learn more About Exporting Data and Images Out of Rocky (p. 1155)

See Also:

- Use the Windows Editors Panel to Change a 3D View (p. 1047)

CHANGE WINDOW DISPLAY AND DATA UPDATE OPTIONS FOR THE SELECTED 3D VIEW

1. Ensure the **Window Editors** panel is visible. (From the **View** menu, click **Window Editors**.)
2. From the Workspace (p. 1396), select the 3D View window that you want to change. (See also View Geometries, Particles, Points, and Fluids in 3D (p. 1041).)
3. From the **Window Editors** panel, select the **3D View** tab and then change the options you want.

Tips:

- You can also open the **Window Editors** panel for a window by right-clicking an empty space within the 3D View window (for example, the background behind the geometries), and then clicking **Settings**.
- You can also use Rocky Preferences to change the background color and fonts for all the 3D View windows in your project, not just selected one. (See the "Preferences for Windows Properties" section in the About Setting Global Preferences (p. 227) topic.)
- To reuse in future projects the settings you made to this tab, click the **Save Current Configuration in Settings** button.

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

- To apply settings you have already saved (either by previously using the **Save Current Configuration in Settings** button or by saving selections within the **Preferences** dialog) to this tab, click the **Restore Configuration from Settings** button.

See Also:

- About Using the Window Editors panel to Change the Selected 3D View (p. 1058)
- About Using the Window Editors Panel to Change All 3D Views in the Project (p. 1048)
- Add or Edit a Text Overlay for a 3D View or Particles Details Window (p. 1069)
- Add or Edit an Image Overlay for a 3D View or Particles Details Window (p. 1070)

ADD OR EDIT A TEXT OVERLAY FOR A 3D VIEW WINDOW

1. Ensure the **Window Editors** panel is visible. (From the **View** menu, click **Window Editors**.)
2. From the Workspace (p. 1396), select the 3D View window that you want to change. (See also View Geometries, Particles, Points, and Fluids in 3D (p. 1041).)
3. From the **Window Editors** panel, select the **Overlays** tab and then do one of the following:
   - To add a new text overlay, click the **Add text overlay** button. A new Text entry appears in the Overlays list at the top of the panel.
   - To edit an existing text overlay, select the name of the text you want to change from the list at the top of the panel.
4. Do any or all of the following:
   - To enter your own text, under **Input**, place your cursor directly into the Text entry and then remove and add text as you like using your keyboard.
   - To have Rocky automatically display simulation-related text, including simulation time or property name, in the **Input** box, place your cursor where you want the text to be inserted, choose what you want from the **Keyword for** list, and then click **Insert**. The placeholder text appears in the **Input** box. Tip:To see the text replaced with real values on the 3D View, click **Apply**.
   - To change how the text is displayed in the window, edit the **Bold**, **Italic**, **Scale Mode**, **Font Size**, **Justification**, and **Color** options as you want. (See also About Using the Window Editors panel to Change the Selected 3D View (p. 1058).)
   - To move the text, do one of the following:
     - Use your mouse to click the center of the text box and then drag it to a new location on the window.
     - Under **Widget**, use the **Position** options for **X** and **Y**.
   - To change the size and shape of the text box, do one of the following:
     - Click the text box border in the 3D View window, and then drag the border to resize or reshape the text box.

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

  - Under **Widget**, use the **Position** options for **Width** and **Height**. Note: While the **Keep aspect ratio** (under **Widget**) setting will allow you to change the shape of the sizing box that surrounds the text, it will not affect the size of the text itself. No matter the shape of the sizing box, Rocky will maximize the text size to fill it.

- To hide the text from the 3D View window, under **Widget**, clear the **Visible** checkbox.
- To lock the size and position of the text box, under **Widget**, clear the **Interact** checkbox.
- To change how transparent the text is, under **Widget**, edit the **Opacity** value. Tip:Click away from the **Opacity** box to have your changes applied to the 3D View.
- To change the appearance of the box surrounding the text, choose what you want under **Widget** for **Border Size** and **Border Color**.
- To remove all the text in the selected overlay, click the **Remove overlay** button.

See Also:

- About Using the Window Editors panel to Change the Selected 3D View (p. 1058)
- Change Window Display and Data Update Options for the Selected 3D View (p. 1068)
- Add or Edit an Image Overlay for a 3D View Window (p. 1070)

ADD OR EDIT AN IMAGE OVERLAY FOR A 3D VIEW WINDOW

1. Ensure the **Window Editors** panel is visible. (From the **View** menu, click **Window Editors**.)
2. From the Workspace (p. 1396), select the 3D View window that you want to change. (See also View Geometries, Particles, Points, and Fluids in 3D (p. 1041).)
3. From the **Window Editors** panel, select the **Overlays** tab and then do one of the following:
   - To add a new image, do all of the following:
     a. Click the **Add image overlay** button. A new Image entry appears in the Overlays list.
     b. Under **Input** click the **...** button next to **Filename**.
     c. From the **Load File** dialog, navigate to and select the image file you want, and then click **Open**.

     The location of the file you selected is displayed in the **Filename** box and the Image entry is appended with the file name. The image you selected appears in the selected window.
   - To edit an existing image, under **Input**, select the entry representing the image that you want to change.
4. Do any or all of the following:
   - To move the image, use your mouse to click the center of the image and drag it to a new location on the window.

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

- To change the size of the image, click the image border in the window, and then drag to resize as desired.

> **Note:**
>
> While the **Keep aspect ratio** (under **Widget**) setting will allow you to change the shape of the sizing box that surrounds the image, it will not affect the size of the image itself. No matter the shape of the sizing box, Rocky will maximize the image size to fill it.

- To hide the image from the window, under **Widget**, clear the **Visible** checkbox.
- To lock the size and position of the image, under **Widget**, clear the **Interact** checkbox.
- To change how transparent the image is, under **Widget**, edit the **Opacity** value. **Tip:** Click away from the Opacity box to have your changes applied to the window.
- To change the appearance of the box bordering the image, choose what you want under **Widget** for **Border Size** and **Border Color**.
- To remove the image from the selected window, click the **Remove overlay** button.

See Also:

- About Using the Window Editors panel to Change the Selected 3D View (p. 1058)
- Change Window Display and Data Update Options for the Selected 3D View (p. 1068)
- Add or Edit a Text Overlay for a 3D View Window (p. 1069)

ABOUT USING THE WINDOW EDITORS PANEL TO CHANGE THE WINDOW AXES DISPLAYS

From the Window Editors panel, you can modify the axes display settings for all 3D View, Motion Preview, and Particles Details windows in your project. The settings you make on the Axes tab for any of the three window types will apply to the other two window types respectively. And any changes you make will also be replicated in future Rocky projects.

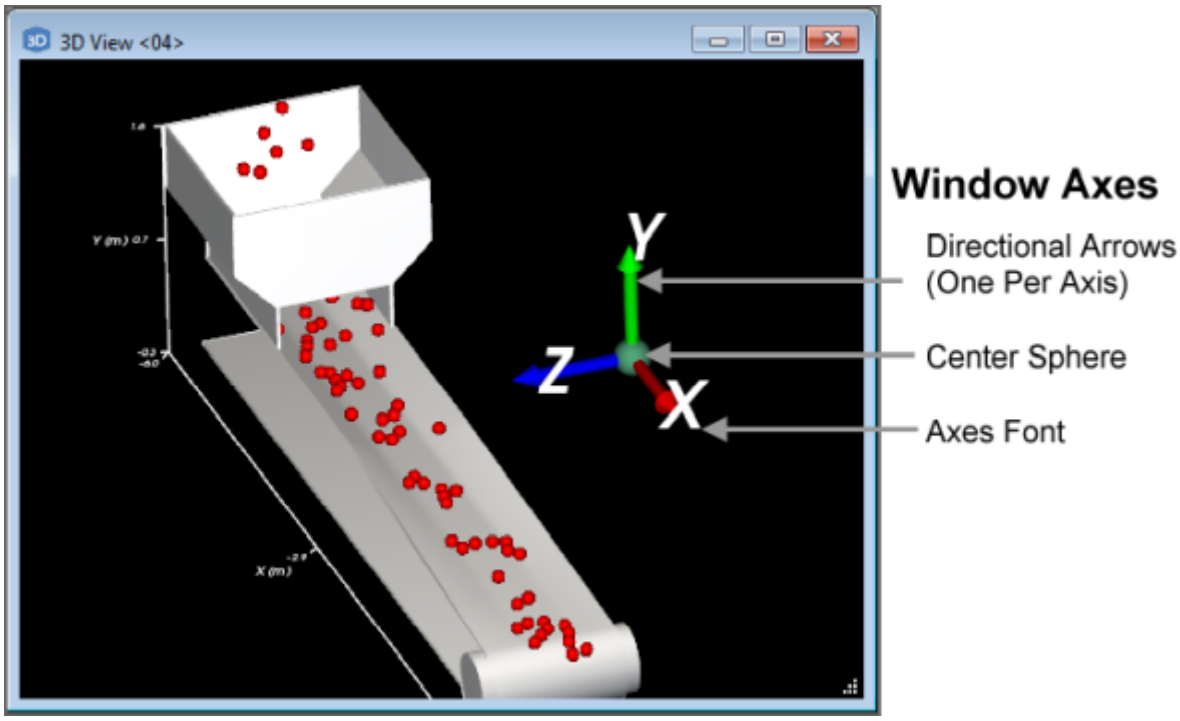
> **Important:**
>
> The axes settings described here apply only to the set of axes representing the window itself. Other axes in Rocky, including the set of axes representing a motion frame and the set of axes representing an inlet, are not affected by these settings.

Axes Display Options

The options on the Axes tab enable you to change the display of the colored axes set on each of your Rocky windows. The settings include changing the individual arrow colors, axes fonts, and the size and placement of the axes set itself:

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

Figure 5.61: Window Axes options that can be changed



AXES SETTINGS ON THE WINDOW EDITORS PANEL

See the images and table below to understand more about how you can use the Window Editors panel to change all Axes settings for the windows in your project.

Figure 5.62: Axes options on the Window Editors panel when the main 3D View entity is selected

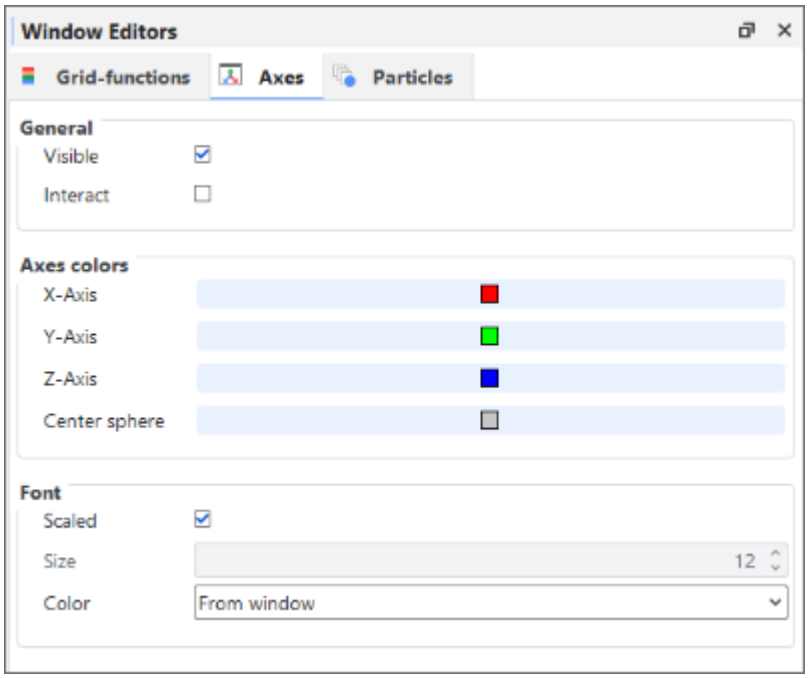


Table 1: Axes options on the Window Editors panel

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

Setting	Description	Range
General		
Visible	When enabled, shows a set of colored axes on all windows in your project.	Turns on or off
Interact	When enabled (and the **Visible** option is enabled as well), this allows you to move and change the size of the colored axes set on all windows in your project.	Turns on or off
Axes colors		
X-Axis	Enables you to set the color of the axis arrow that represents the X coordinate direction.	Limited by the options available in the **Select Color** dialog.
Y-Axis	Enables you to set the color of the axis arrow that represents the Y coordinate direction.	Limited by the options available in the **Select Color** dialog.
Z-Axis	Enables you to set the color of the axis arrow that represents the Z coordinate direction.	Limited by the options available in the **Select Color** dialog.
Center sphere	Enables you to set the color of the sphere at the center of the axes.	Limited by the options available in the **Select Color** dialog.
Font		
Scaled	When enabled, the size of the text used on the set of axes will be determined in relation to the height and width of the window itself; the axes text will grow or shrink in scale with the size of the window. When cleared, the font size can be set through the **Size** text box.	Turns on or off
Size	The value entered here determines the size of the axes text in relation to the computer screen. With this option, the text (and its sizing box) stays the same size no matter the height and width of the window that contains it.	Whole positive values
Color	Defines how the font color of the axes text is determined. Specifically: • **From window** uses the same font color as defined for the window itself. • **Custom** enables you to select a different color for only the text used with the axes.	From window; Custom

What would you like to do?

- Change the Axes Display for all Rocky Windows (p. 1074)

See Also:

- About Using the Window Editors Panel to Change All 3D Views in the Project (p. 1048)

CHANGE THE AXES DISPLAY FOR ALL ROCKY WINDOWS

1. Ensure that both the **Windows** and **Window Editors** panels are visible. (From the **View** menu, click **Windows**. From the **View** menu, click **Window Editors**.)
2. From the **Windows** panel, select the top-most item for any of the three window categories. This includes the **3D View**, **Motion Preview**, and **Particles Details** entities.
3. From the **Window Editors** panel, on the **Axes** tab, change the options you want. The changes you make are replicated to all windows in the project and will also be retained for future projects.

Important:

The axes settings described here apply only to the set of axes representing the window itself. Other axes in Rocky, including the set of axes representing a motion frame and the set of axes representing an inlet, are not affected by these settings.

See Also:

- About Using the Window Editors Panel to Change the Window Axes Displays (p. 1071)

Using the Coloring Tab to Change a 3D View

The Coloring tab, located on the Data Editors panel for all simulation entities and User Processes, enables you to change the colors and data attributes of the entities being displayed in a 3D View window. Faces, edges, nodes, and vectors can be shown or hidden; displayed in solid colors; or colored by a Property property of your choice.

In addition, the Coloring tab also enables you to modify the limits, units, and legend options for the color scales used in your view windows.

For quicker access in 3D View windows only, some of these same options are also available on the Coloring toolbar.

What would you like to do?

- Learn more About Using the Coloring Tab to Change a 3D View (p. 1074)
- Learn more About Color Scales (p. 1086)
- Learn more About Using the Coloring Toolbar to Change a 3D View (p. 1084)
- Use the Coloring Tab to Change a 3D View (p. 1092)
- Use the Coloring Toolbar to Change a 3D View (p. 1093)

See Also:

- Change a 3D View Window (p. 1042)

ABOUT USING THE COLORING TAB TO CHANGE A 3D VIEW

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

Using the options on the Coloring tab is one way you can change what is displayed in a 3D View window for the simulation entity (p. 1395) that you want to change. Visualization options for the Faces, Edges, and Nodes can be modified with the Coloring tab for any of the following entities:

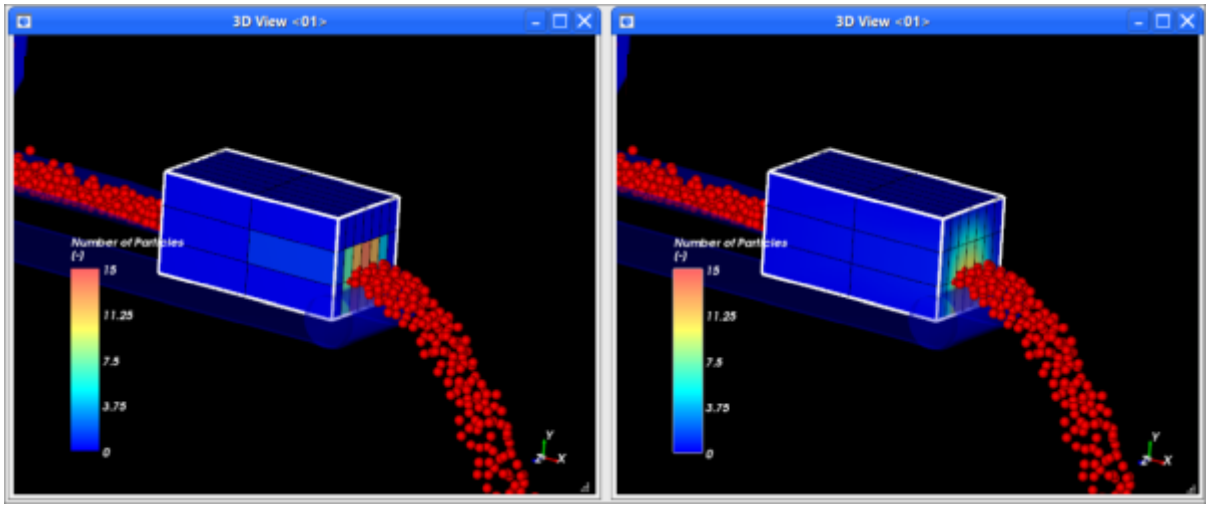
- Default Conveyors or Imported Geometry Components (p. 480)
- Inlets
- Particles (p. 602)
- Point Clouds (p. 699)
- Contacts (p. 746)
- 1-Way LBM (p. 771)
- 1-Way Fluent (p. 775)
- 2-Way Fluent (p. 795)
- Particles Trajectories User Process (p. 1209)
- Eulerian Statistics (p. 1226)
- Non-Particle-Specific User Processes, which includes a Cube (p. 1176), Cylinder (p. 1181), Polyhedron (p. 1186), Plane (p. 1195), Property (p. 1202), or Cell Inspector (p. 1211) User Process
- Any other User Process created from any of the above entities

For Color scales, there is also a Coloring tab but the options presented are quite different than the ones explained below. For more information, see also About Color Scales (p. 1086).

EXAMPLE EFFECTS

Example effects you can achieve with this method include smoothing out the Faces visualization of an Eulerian Statistics property by enabling **Show on Nodes**:

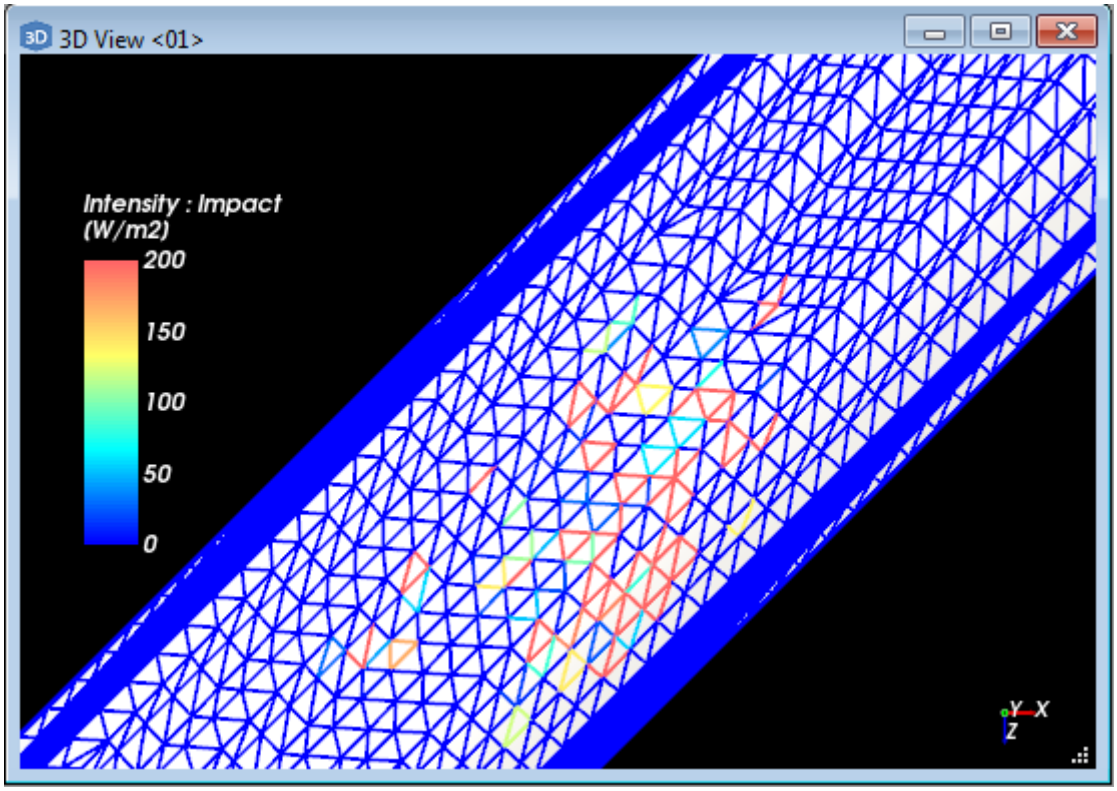
Figure 5.63: A Cube User Process with Eulerian Statistics applied displays Number of Particles with Show on Nodes disabled (left) and enabled (right)



Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

For default conveyors and imported geometry components, you can also color Edges by a property (Figure 5.64: A custom conveyor showing Solid Color Faces with Edges showing Intensity : Impact (p. 1076)); enlarge the point size of Nodes (Figure 5.65: A custom conveyor showing 70% Transparent Solid Color Faces with increased Point Size Nodes showing Intensity : Impact (p. 1077)); and smooth out the Faces visualization of a property by enabling **Show on Nodes** (Figure 5.66: A custom conveyor with Faces showing Intensity : Impact with Show on Nodes disabled (left) and enabled (right) (p. 1077)).

Figure 5.64: A custom conveyor showing Solid Color Faces with Edges showing Intensity : Impact



Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

Figure 5.65: A custom conveyor showing 70% Transparent Solid Color Faces with increased Point Size Nodes showing Intensity : Impact

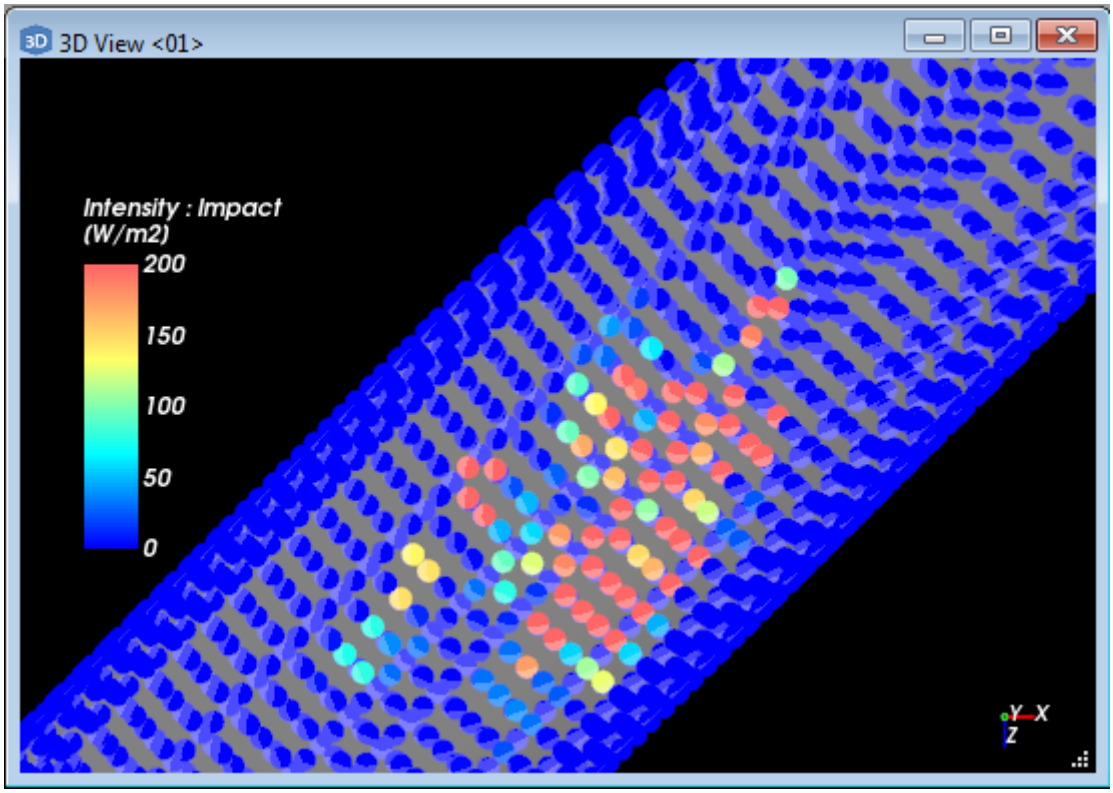
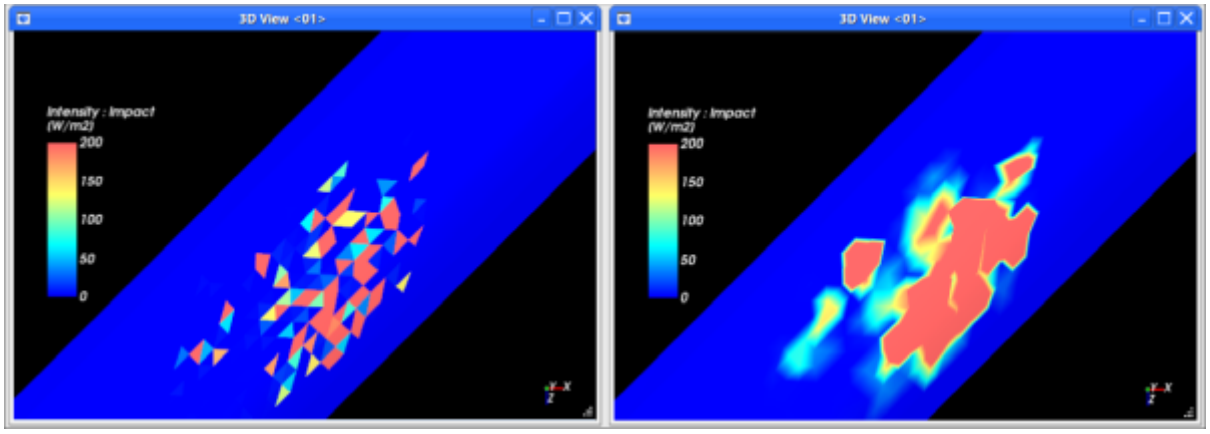


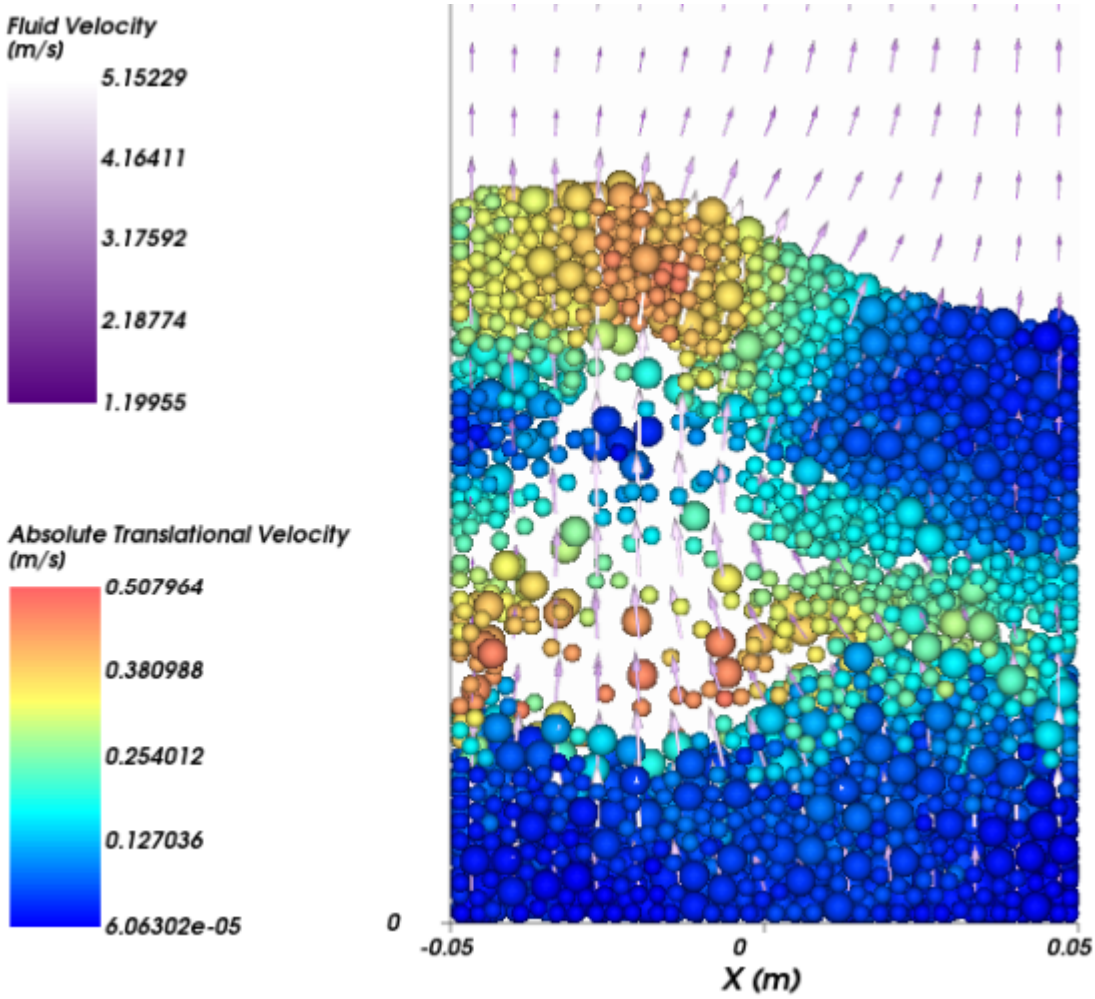
Figure 5.66: A custom conveyor with Faces showing Intensity : Impact with Show on Nodes disabled (left) and enabled (right)



For CFD Coupling, you can also visualize the flow of fluids by viewing them as Nodes or Vectors. In 2-Way Fluent, for example, the fluid behavior is transient so as the simulation progresses, you can better see how the fluid flow interacts with the particle flow and vice versa:

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

Figure 5.67: Example of a 2-Way Fluent simulation showing transient fluid flow vectors (in purple), and transient particle nodes (all colors)

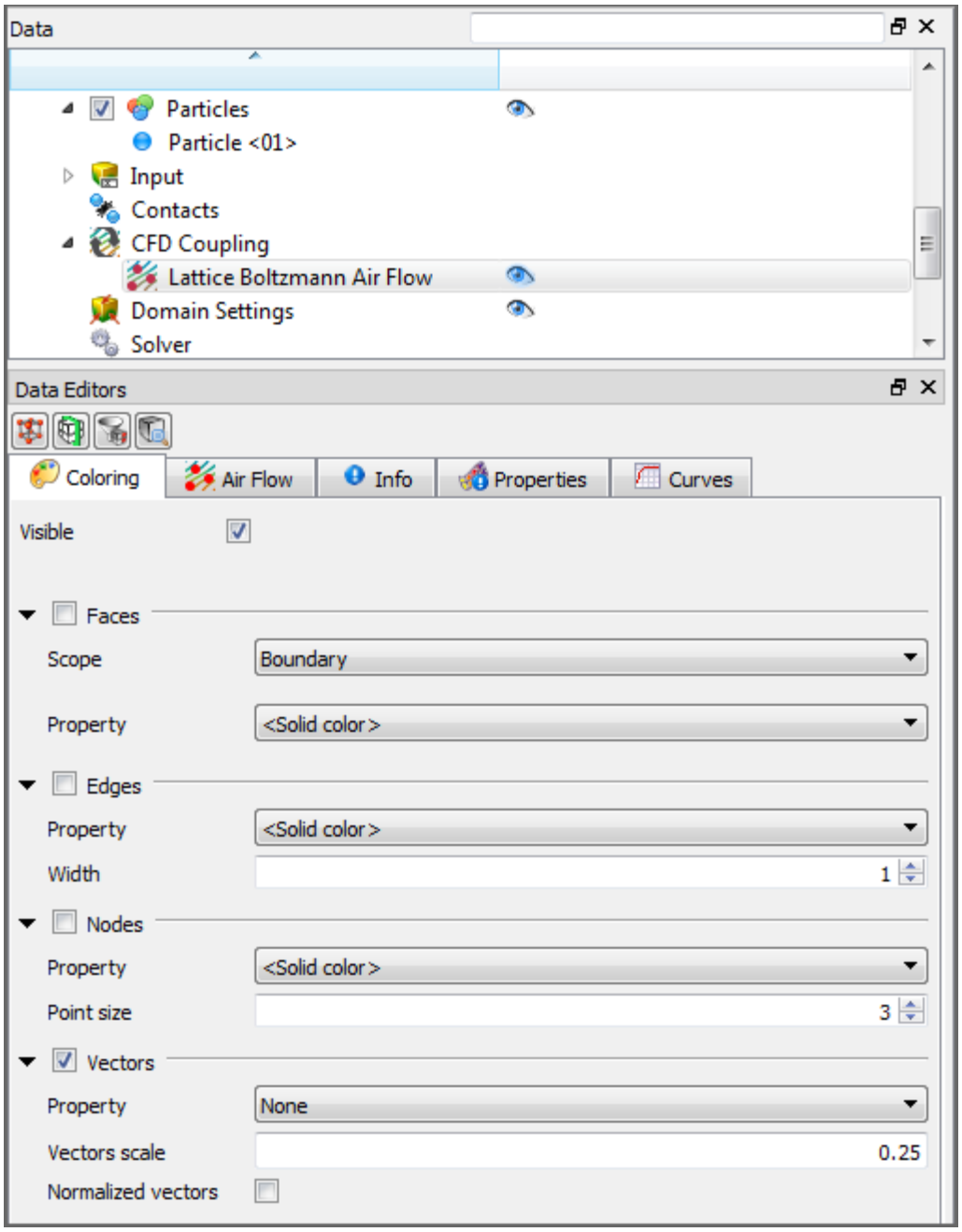


PARAMETER DEFINITIONS

See the image and table below to help you understand how to use the Coloring tab to change a 3D View for selected entity.

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

Figure 5.68: Example Coloring tab in the Data Editors panel showing 1-Way LBM



Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

Figure 5.69: Coloring tab in the Data Editors panel for a Point Cloud

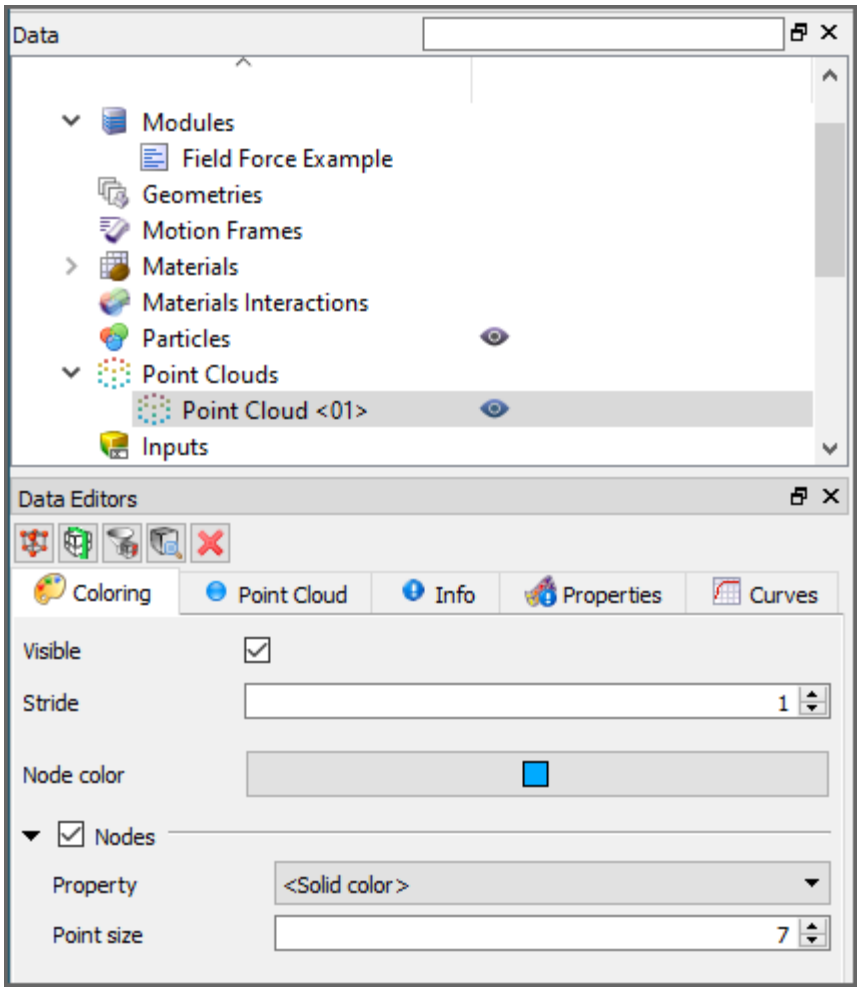


Table 1: Coloring tab parameter options (all entities except Color scales)

Setting	Description	Range	Applicable Entities
Visible	When enabled, shows the selected entity in the active view window. **Note:** This is the same as using the Data panel eye icon. (See also Show/Hide Components by Using Eye icons and Checkboxes.)	Turns on or off	All entities
Stride	One out of this number of data points will be shown when displaying Nodes or Vectors. The lower the number, the more data points will be displayed. Choosing a higher number	Whole numbers 1 or greater	Particles (Vectors only); Point Clouds; Contacts;

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

Setting	Description	Range	Applicable Entities
	can help speed up displays with very large amounts of data. **Example:** Choosing a Stride of 5 means that 1 point out of every 5 will be shown in the 3D View.		1-Way Fluent; 2-Way Fluent
Transparency	Enables boundaries to be drawn transparently in the selected view window. A value of 0 (zero) makes the faces completely solid or opaque. A value of 100 makes the faces completely transparent or hidden.	Turns on or off; 0-100	Default Conveyors; Imported Geometry Components; Eulerian Statistics
Color	When **Faces**, **Edges**, and/or **Nodes** are selected and the **Property** is <Solid Color>, this enables the selected display type to be colored in the one solid color chosen. Note: For **Faces**, **Transparency** options affect how opaque this color appears in the view.	Options limited by the **Select Color** dialog	All entities
Faces	Enables the entire surface of the entity to be visible and affected by the various Faces coloring options.	Turns on or off	Default Conveyors; Imported Geometry Components; 1-Way LBM; Eulerian Statistics
Scope	When **Faces** is enabled, this changes how the blocks or cells are drawn in the view. Specifically: • **Boundary** colors only the outer surface of the blocks or cells. • **Full** extends the coloring to the entire block or cell. Note that this coloring is only perceptible when **Transparency** is enabled.	Full; Boundary	1-Way LBM; Eulerian Statistics
Structured Partitions	When **Faces** is enabled, this determines what side(s) of each block is visible. Specifically: • **All** keeps all sides of the blocks visible. • **Custom** enables you to set which side(s) you want to show.	All; Custom	Eulerian Statistics

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

Setting	Description	Range	Applicable Entities
Structured Custom Partitions	When **Faces** is enabled and **Custom** is chosen from the **Structured Partitions** list, this enables you to select one or more sides that you want visible on each block.	Top; Bottom; North; South; East; West;	Eulerian Statistics
Property	Three separate lists under **Faces**, **Edges**, and **Nodes** respectively provide properties and other color options that apply to the display type within which the list is contained. Specifically: • **<Solid Color>** applies a single color to the display type. • The remaining properties come directly from the component's Properties tab. Selecting one of these properties colors display type in a range according to the data collected for the current time step. (See also About Properties (p. 933)).	<Solid color>; List of properties automatically generated from the Properties tab	All entities
Show on Node?	When **Faces** is enabled and a property is selected from the **Property** list under **Faces**, this enables the visualization of the selected property to be evenly interpolated across the entity for a smoother visualization.	Turns on or off	Default Conveyors; Imported Geometry Components; 1-Way LBM; Eulerian Statistics
Edges	Enables only the edges bordering the components making up the entity boundaries visible and affected by the various Edges coloring options.	Turns on or off	Default Conveyors; Imported Geometry Components; Particle Trajectories; 1-Way LBM; Eulerian Statistics; Contacts
Width	When **Edges** is selected, this changes the thickness of the lines used to draw the borders. Entering 1 will provide the narrowest line.	Positive values	Default Conveyors; Imported Geometry Components; Particle Trajectories;

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

Setting	Description	Range	Applicable Entities
			1-Way LBM; Eulerian Statistics; Contacts
Nodes	Enables the points marking the intersections of the entity's individual components to be visualized as dots and affected by the various Nodes coloring options.	Turns on or off	All entities
Level of Detail	Determines how the Particle visualization will be rendered. The more detailed the rendering, the more time it will take to visualize. Specifically: • **Rich** renders particles with as much detail as possible but takes the most time and resources to visualize. • **Automatic** enables Rocky to determine whether the particles are richly or simply rendered based both upon the time taken to visualize and whether the user is currently manipulating the camera. • **Simple** renders the particles with as little detail as possible but takes the least time and resources to visualize.	Rich; Automatic; Simple	Particles
Point Size	When **Nodes** is selected, this changes the size of the dots used to draw the nodes. Entering 1 will provide the smallest dot.	Positive values	All entities
Vectors	Enables the entity's data to be visualized as vectors and be affected by the various Vectors coloring options.	Turns on or off	Particles; Particle Trajectories; 1-Way LBM; 1-Way Fluent; 2-Way Fluent; Contacts
Vectors Scale	When **Vectors** is selected and a property chosen from the **Property** list under **Vectors** (not **None**), this changes the scale of the vectors drawn. The lower the number, the smaller the vectors will be displayed. Unless **Normalized Vectors** is enabled, the relative size of the vectors is based upon the property being displayed.	Positive values	Particles; Particle Trajectories; 1-Way LBM; 1-Way Fluent; 2-Way Fluent; Contacts
Normalized Vectors	When **Vectors** is selected, this enables the vectors to be drawn all the same size according to the **Vectors Scale** value.	Turns on or off	Particles; Particle Trajectories; 1-Way LBM;

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

Setting	Description	Range	Applicable Entities
			1-Way Fluent; 2-Way Fluent; Contacts

What would you like to do?

- Use the Coloring Tab to Change a 3D View (p. 1092)
- Learn more About Using the Coloring Toolbar to Change a 3D View (p. 1084)

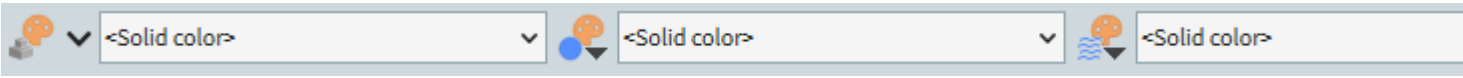
See Also:

- About Using the Coloring Tab to Change the Motion Preview for a Frame (p. 579)
- About Using the Coloring Tab to Change the Preview for a Particle Set (p. 695)
- About Using the Coloring Tab to Change a Plot or Histogram (p. 1152)

ABOUT USING THE COLORING TOOLBAR TO CHANGE A 3D VIEW

The Coloring toolbar (shown below) is the method by which you can quickly change visualization options for a 3D View that affects all geometries at once, or all particles at once, or all SPH elements — all from one central location.

Figure 5.70: Coloring toolbar

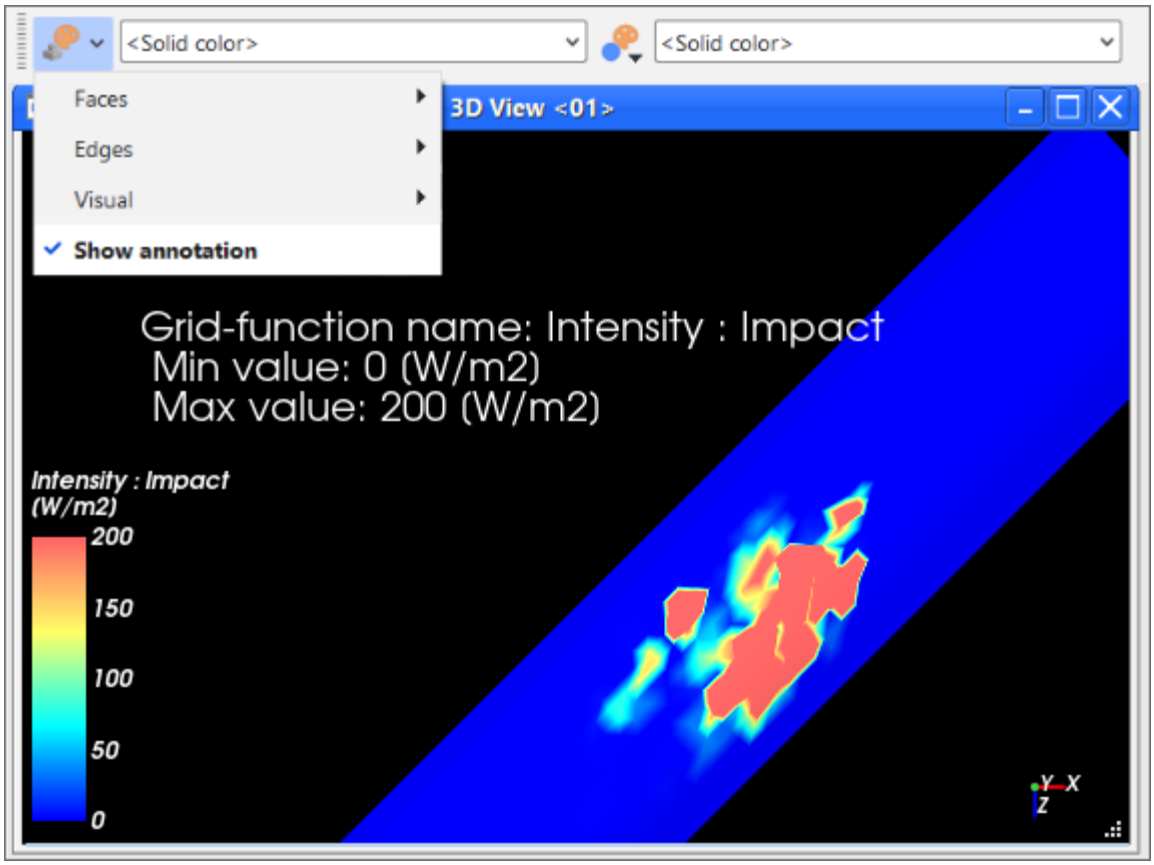


The coloring toolbar presents some of the same options as the Coloring tab does for individual geometry components, Particles or SPH elements (see also About Using the Coloring Tab to Change a 3D View (p. 1074)). These settings include what Property to display in the 3D View, as well as color and display options for Faces, Edges, and Nodes.

For geometries, you can also use the **Show Annotation** option to add a quick text overlay with default values:

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

Figure 5.71: Show Annotations provides a default text overlay for geometries



PARAMETERS AND SETTINGS

See the images and table below to help you understand how to use the Coloring toolbar to change a 3D View.

Table 1: Coloring toolbar button descriptions

Button / Menu Option	Description
Coloring settings (Geometries)	Presents a menu of options, including Faces, Edges, and Visual, as well as Annotation display options. Options selected here affect all geometries within the project.
Coloring settings (Particles)	Presents a menu of options, including Visible, Color, and Visual display options. Options selected here affect all Particles within the project.
Coloring settings (SPH Elements)	Presents a menu of options, including Visible and Color options. Options selected here affect all SPH Elements within the project.
<Solid color> Property list	Two separate lists—one affecting all geometries and one affecting all Particles in the project— provide properties and other color options for the respective *entity* (p. 1395). Specifically: • <Solid Color> applies a single color to the entity.

Button / Menu Option	Description
	• The remaining properties come directly from the component's Properties tab. Selecting one of these properties colors the entity in a range according to the data collected for the current time step. (See also About Properties (p. 933)).
Faces	Settings for this display type apply to the entire surface of the entity.
Edges	Settings for this display type apply only to the edges bordering the components making up the entity boundaries.
Visible	When enabled, shows the selected display type or entity in the active 3D View window. Note: This is the same as using the Data panel eye icon. (See also Show/Hide Components by Using Eye icons and Checkboxes (p. 113)).
Color	When the Property is <Solid Color>, this enables the selected display type or entity to be colored in the one solid color chosen. Note: For Faces, Transparency options affect how opaque this color appears in the 3D View.
Visual	Specifies the level of detail in which the entity is rendered. Specifically: • **Point** renders the entity with as little detail as possible but takes the least time and resources to visualize. • **Solid/Point** enables Rocky to determine automatically whether the entity is rendered richly (Solid) or simply (Point) based both upon the time taken to visualize the entity and whether the user is currently manipulating the camera. • **Solid** renders the entity with as much detail as possible but takes the most time and resources to visualize.
Show annotation	For geometries only, creates a new text overlay showing the Property name, and both the minimum and maximum values (Figure 5.71: Show Annotations provides a default text overlay for geometries (p. 1085)). These default values can be changed by editing the overlay. (See also Add or Edit a Text Overlay for a 3D View Window (p. 1069)).

What would you like to do?

- Use the Coloring Toolbar to Change a 3D View (p. 1093)

See Also:

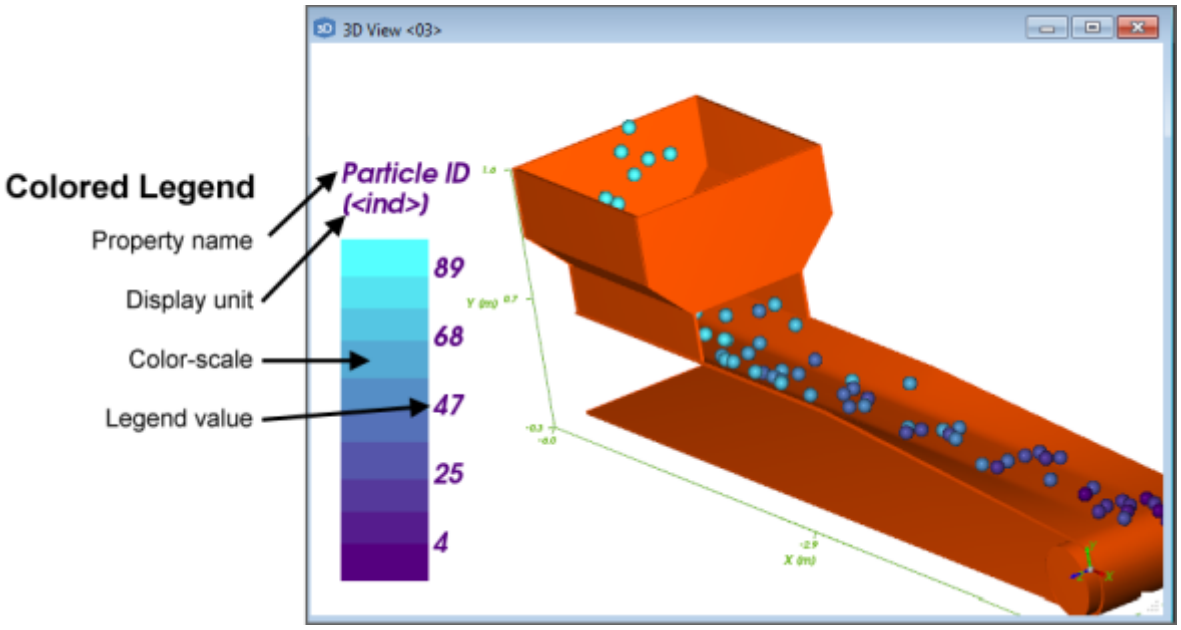
- Change a 3D View Window (p. 1042)

ABOUT COLOR SCALES

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

Every time you choose to show a *Property* (p. 1394) in a 3D View or Particles Details window, a new colored legend is added to help you understand the data values shown:

Figure 5.72: Color Scale options that can be changed on the Coloring tab



For each colored legend created, a new entry is added under **Color Scales** in the **Data** panel. By selecting the entry representing it under **Color Scales**, you can change the appearance of a colored legend—including the colors making up the banding, the number of value ticks, and the units displayed for the legend values—by modifying the options on the **Coloring** tab (Figure 5.73: Coloring tab options for Color Scale entry in the Data Editors panel (p. 1088)).

The options you change apply to all 3D View and Particles Details windows you add to your project.

COLOR SCALE SETTINGS

Use the images and tables below to help you understand how to modify your 3D View and Particles Details Legends using Color Scales.

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

Figure 5.73: Coloring tab options for Color Scale entry in the Data Editors panel

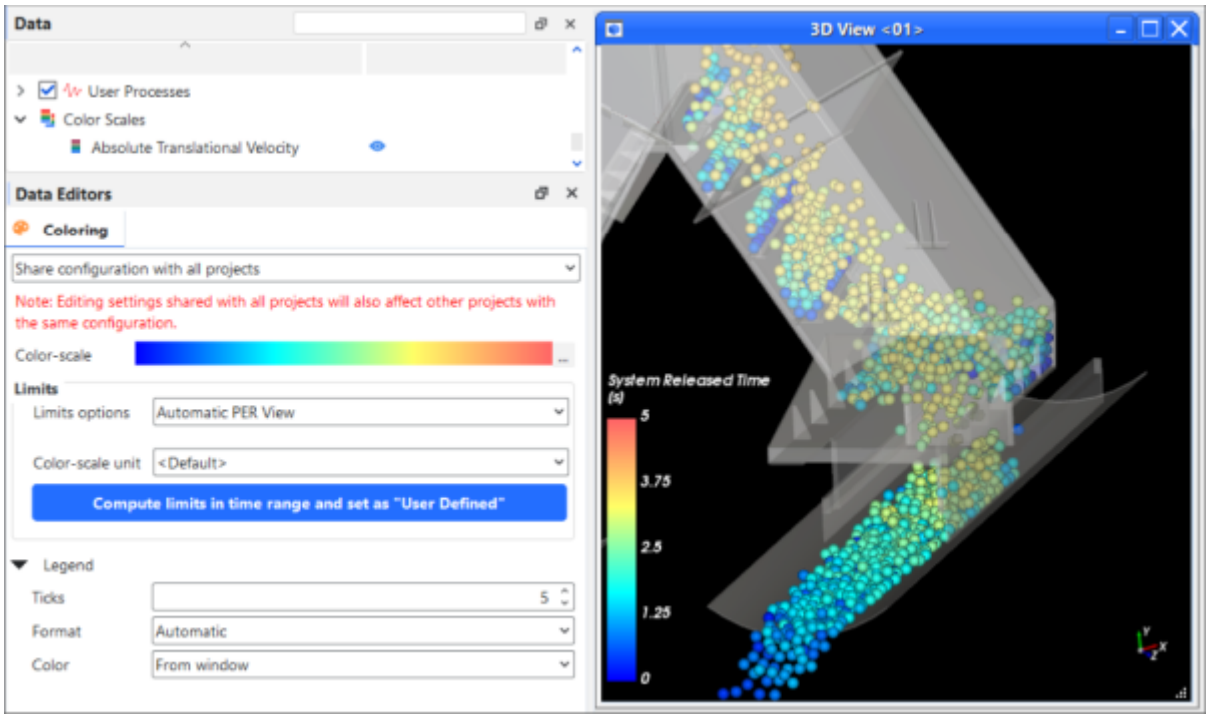


Table 1: Coloring tab options for Color Scale entries in the Data Editors panel

Setting	Description	Range
Share configuration with all projects	When **Share configuration with all projects** is selected, any changes you make to the **Color-scale** or its **Limits** will be applied to all Rocky projects that both uses the selected Color Scale and also has the **Share configuration with other projects** setting selected.	Share configuration with all projects; Save changes only for current project
Save changes only for current project	When **Save changes only for current project** is selected, any changes you make to the **Color-scale** or its **Limits** will apply only to the current project whenever the selected Color Scale is used.	Share configuration with all projects; Save changes only for current project
Color-scale	Displays the current color banding configuration for the selected legend. Clicking the **...** button will enable you to edit the options. (See Figure 5.74: Color-scale dialog (p. 1090)and Table 2 below.)	Display only
Limits		
Limits options	Determines how the minimum and maximum values for the selected legend are assigned. Specifically: • **Automatic PER view** enables Rocky to determine, for the selected window only, the limit values automatically based upon the current time step.	Automatic PER view; Automatic ALL views; User Defined

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

Setting	Description	Range
	• **Automatic ALL views** enables Rocky to determine, for all windows using the selected Color Scale, the limit values automatically based upon the current time step. • **User Defined** enables you to manually set the limit values. **Tip:** You may also have Rocky help you calculate the limits by specifying a time range based upon an applicable process(es). To do so, click the **Compute limits in time range and set as "User Defined"** button. (See also Figure 5.75: Calculate limits for time range dialog (p. 1091) and Table 3 below.)	
Limits	When **Limit options** is set to **User Defined**, this enables you to enter the minimum and maximum values you want to define the legend's limits.	Any value
Color-scale unit	Determines the units that will be displayed in the legend. Note: If the **Color-scale unit** differs from the unit you selected in the **Limits** drop-down list, your limit values will be translated into this preferred unit on the actual legend display.	Options limited by list box
Legend		
Ticks	Defines the number of divisions whose respective values will be displayed next to the legend. **Note:** Ticks represent only the numerical divisions of the legend. To change the color divisions, use the Banded option on the Color-scale dialog. (See Table 2 below.)	1 > 65
Format	Enables you to change how the values next to each tick are formatted. Specifically: • **Integer** displays full values without decimals. • **Decimal** options display the specified number of digits following the decimal. • **Exponential** options display the specified number of digits prior to the exponent.	Integer; Decimal 2 digits; Decimal 4 digits; Decimal 8 digits; Exponential 2 digits; Exponential 4 digits; Exponential 8 digits; Automatic

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

Setting	Description	Range
	• **Automatic** displays the Rocky default format for the values.	
Color	Defines how font color of the values listed next to the legend is determined. Specifically: • **From window** uses the same font color as defined for the selected window itself. • **Custom** enables you to select a different color for only the values displayed next to the legend.	From window; Custom

Figure 5.74: Color-scale dialog

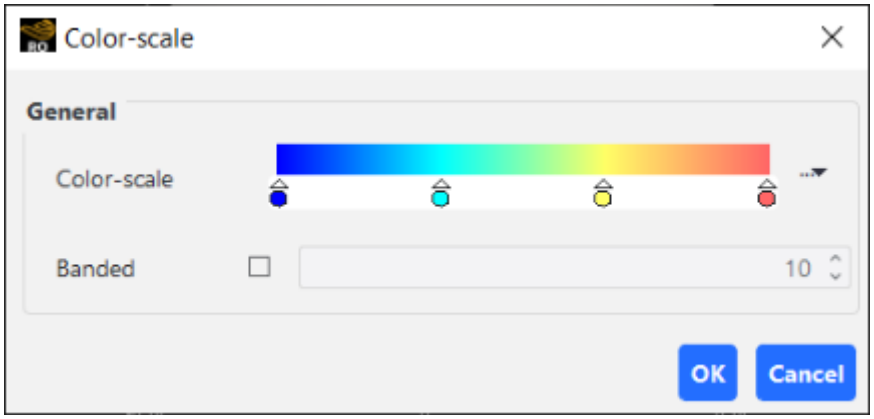


Table 2: Color-scale dialog options

Setting	Description	Range
Color-scale	Determines the gradient distribution of color along the selected scale. From here, you can move the color sliders to adjust the gradient, double-click color slider handles to change the color, double-click the slider bar itself to add another color slider, or click the **...** button to select from several additional presets, which you can also modify.	Limited by the UI
Banded	When selected, the legend will divide the color scale into the number of solid bands you indicate. When cleared, the legend will display the color scale in a smooth, unbanded gradient. **Note:** Banded applies only to the color divisions of a legend. To change the numerical divisions, change the Ticks value instead. (See Table 1 above.)	Turns on or off; Positive whole numbers 2 or greater

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

Figure 5.75: Calculate limits for time range dialog

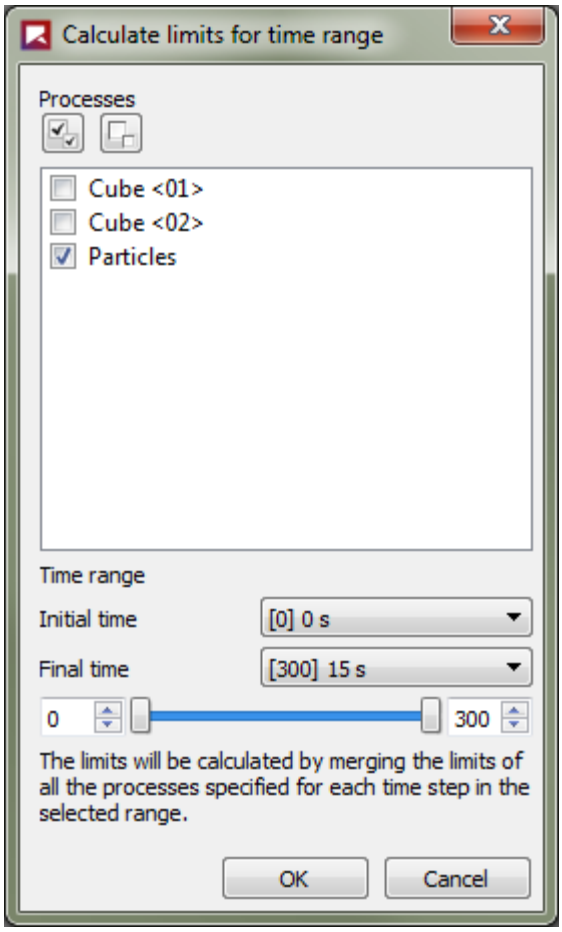


Table 3: Calculate limits for time range dialog options

Setting	Description	Range
Processes	Lists the applicable *entities* and User Processes upon which you can base your time range. You must select at least one process to calculate limits using a time range.	Limited by the UI
Time range		
Initial time	The time step with which to begin the time range. This can be set by selecting the time step you want from the drop-down list, by entering the corresponding output number for the time step, or by using the slider.	Limited by the UI
Final time	The time step with which to end the time range. This can be set by selecting the time step you want from the drop-down list, by entering the corresponding output number for the time step, or by using the slider.	Limited by the UI

What would you like to do?

- Change Color Scale Options for 3D View or Particles Details Legends (p. 1093)

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

- Learn more About Using the Window Editors Panel to Change All 3D Views in the Project (p. 1048)

See Also:

- Using the Coloring Tab to Change a 3D View (p. 1074)
- About Using the Coloring Tab to Change the Preview for a Particle Set (p. 695)

USE THE COLORING TAB TO CHANGE A 3D VIEW

1. From the Workspace (p. 1396), select the 3D View window that contains the simulation entity (p. 1395) for which you want to change the visualization.
2. From the **Data** panel, select the entity that you want to change. This can include any of the of the following items:
   - Default Conveyors or Imported Geometry Components
   - Inlets
   - Particles
   - Point Clouds
   - Contacts
   - 1-Way LBM
   - 1-Way Fluent
   - 2-Way Fluent
   - Particles Trajectories User Process
   - Eulerian Statistics User Process
   - Non-Particle-Specific User Processes, which includes a Cube, Cylinder, Polyhedron, Plane, Property, or Cell Inspector User Process
   - Color Scales

   Tip:

   You can also apply settings to multiple, common items at once by multi-selecting items in the **Data** panel. Only those settings common to the items you have chosen will be shown in the **Data Editors** panel.

3. From the **Data Editors** panel, select the **Coloring** tab, and then change the settings you want. The visualization settings you change are automatically updated in the selected view window.

See Also:

- About the Coloring Tab (p. 93)

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

- Using the Coloring Tab to Change a 3D View (p. 1074)

CHANGE COLOR SCALE OPTIONS FOR 3D VIEW OR PARTICLES DETAILS LEGENDS

1. From the Workspace (p. 1396), select a 3D View or Particles Details window containing the legend you want to change.
2. From the **Data** panel, under **Color Scales**, select the entry representing the legend you want to change. Tip:The entry(s) applicable to the selected window will have an open eye icon displayed to the right of the label.
3. From the **Data Editors** panel, on the **Coloring** tab, modify the settings you want. The changes you make are reflected in all 3D View and Particles Details windows that make use of that legend.

See Also:

- About Color Scales (p. 1086)
- Using the Coloring Tab to Change a 3D View (p. 1074)
- Using the Coloring Tab to Change a Particles Details Window (p. 694)
- About Using the Window Editors Panel to Change All 3D Views in the Project (p. 1048)

USE THE COLORING TOOLBAR TO CHANGE A 3D VIEW

1. Select the 3D View that you want to change.
2. From the **Coloring** toolbar, do one or both of the following:
   - Change the display options for all geometries in the project by making selections from the **Coloring setting**  menu button and corresponding Property  list.
   - Change the display options for all Particles in the project by making selections from the **Coloring setting**  menu button and corresponding Property  list.

See Also:

- About Using the Coloring Toolbar to Change a 3D View (p. 1084)
- Change a 3D View Window (p. 1042)

5.2.4. Save a View with Custom Cameras

The options on the Custom Presets toolbar (shown below) enable you to save the camera visualization settings—which include rotation (tilt), magnification (zoom), and location (pan)—you have made for the selected 3D View, Motion Preview, or Particles Details window. By saving these settings as camera

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

presets, you can always return a view to the preset you prefer or can replicate views for other view windows within your current project.

Figure 5.76: Custom Presets toolbar



What would you like to do?

- Learn more About the Custom Presets Toolbar (p. 1094)
- Save Custom Camera Settings for a View (p. 1096)
- Apply a Custom Camera Preset to a View (p. 1096)
- Modify or Remove an Existing Custom Camera Preset (p. 1097)

See Also:

- Create and Modify a 3D View Window (p. 1040)
- Create and Modify a Motion Preview Window (p. 571)
- Create and Modify a Particles Details Window (p. 679)

About the Custom Presets Toolbar

The Custom Presets toolbar (Figure 5.76: Custom Presets toolbar (p. 1094)) enables you to save camera presets for your 3D View, Motion Preview, and Particles Details windows.

WHAT CUSTOM PRESETS INCLUDE

A preset saves the rotation (tilt), magnification (zoom), and location (pan) settings you have made for the view window you select. These kinds of camera settings can be changed in the following two ways:

- Using the Camera Visualization Toolbar (p. 1045)
- Using the Mouse and Keyboard (p. 1043)

By saving these settings as camera presets, you can always return the position of a view to the preset you prefer or you can replicate views for other view windows within your current project. Custom presets can also help you save time when setting up views for your simulation animations. (See also About Creating and Saving an Animation (p. 1166).)

WHAT CUSTOM PRESETS DO NOT INCLUDE

A preset does not save the window display settings (such as the background color or font sizes) nor the object display settings (such as the transparency, Faces, or Edges treatments) that you have set for the window. Rather, it saves only the relative position of the object being displayed within the window.

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

In addition, presets are not constrained to the type of object being shown in the window, nor even the type of window itself.

For example, you can save a preset that you made of a particle shape within a Particles Details window and apply that preset to a geometry in a Motion Preview window. However, because the view of the particle shape was likely much more magnified due to the smaller size of the particle, the same view applied to the Motion Frame is likely to be too magnified to fully contain the geometries within the view.

Therefore, when saving and applying custom presets across different kinds of view windows, you may need to "Fit" the view after applying the preset. Tip: Press the **R** key on your keyboard to **Fit** the active components into view.

CUSTOM PRESET TOOLBAR SETTINGS

See the images and table below to help you understand how to use the Custom Presets toolbar to save, apply, and modify camera presets for your view windows.

Figure 5.77: Custom Presets toolbar



Figure 5.78: Create new preset dialog

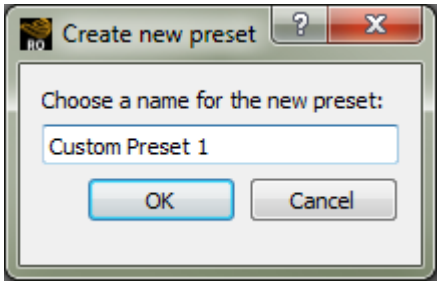


Table 1: Custom Presets toolbar button descriptions

Button	Description
Add preset	Enables you to add a new custom preset to the list. Clicking this button opens the **Create new preset** dialog (shown above).
Select a Preset Select a Preset	Presents a list of presets you have saved. Selecting an item from this list will apply the saved settings to the selected view window.
Update selected preset	Enables you to update the currently selected preset with whatever camera changes that you made to the view. This, in effect, saves over the current preset with the new camera settings.
Remove selected preset	Enables you to remove the selected preset from the list.

What would you like to do?

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

- Save Custom Camera Settings for a View (p. 1096)
- Apply a Custom Camera Preset to a View (p. 1096)
- Modify or Remove an Existing Custom Camera Preset (p. 1097)

See Also:

- Create and Modify a 3D View Window (p. 1040)
- Create and Modify a Motion Preview Window (p. 571)
- Create and Modify a Particles Details Window (p. 679)

Save Custom Camera Settings for a View

1. Select the 3D View, Motion Preview, or Particles Details window displaying the camera visualization settings you want to save. (See also Change a 3D View Window (p. 1042), Change a Motion Preview Window (p. 575), or Change a Particles Details Window (p. 682).)
2. From the **Custom Presets** toolbar, click the **Add preset** button.
3. From the **Create new preset** dialog, enter a name for the preset, and then click **OK**. The new preset is added to the Custom Cameras list.

See Also:

- About the Custom Presets Toolbar (p. 1094)
- Apply a Custom Camera Preset to a View (p. 1096)
- Modify or Remove an Existing Custom Camera Preset (p. 1097)

Apply a Custom Camera Preset to a View

1. Select the 3D View, Motion Preview, or Particles Details window to which you want to apply an existing Custom Camera Preset. (See also Save Custom Camera Settings for a View (p. 1096).)
2. From the **Custom Presets** toolbar, select the name of the preset you want from the Custom Cameras drop-down list. The selected view window is updated to reflect the visualization and camera settings saved in the preset you selected.

Tips:

- If applying a preset to a window type other than the one for which the preset was originally saved, you may need to "Fit" the view after applying the preset to best see the changes. (Press the **R** key on your keyboard to **Fit** the active components into view.)
- If the preset you want is already active in the Custom Cameras list, select first another item in the list (for example, the default **Select a Preset** option), and then select the preset you want.

See Also:

- About the Custom Presets Toolbar (p. 1094)

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

- Save Custom Camera Settings for a View (p. 1096)
- Modify or Remove an Existing Custom Camera Preset (p. 1097)
- Create and Modify a 3D View Window (p. 1040)
- Create and Modify a Motion Preview Window (p. 571)
- Create and Modify a Particles Details Window (p. 679)

Modify or Remove an Existing Custom Camera Preset

1. From the Workspace (p. 1396), select a 3D View, Motion Preview, or Particles Details window.
2. From the **Custom Presets** toolbar, select the name of the preset you want to modify or remove from the list. The selected view window is updated to reflect the visualization and camera settings saved in the preset you selected.
3. Do one of the following:
   - To modify the settings, make the camera visualization changes you want to the selected view window, and then from the **Custom Presets** toolbar, click the **Update selected preset** button. The camera visualization settings for the preset you selected will be saved over with the new changes.
   - To remove the preset, click the **Remove selected preset** button. The selected preset is removed from the list.

See Also:

- About the Custom Presets Toolbar (p. 1094)
- Save Custom Camera Settings for a View (p. 1096)
- Apply a Custom Camera Preset to a View (p. 1096)
- Create and Modify a 3D View Window (p. 1040)
- Create and Modify a Motion Preview Window (p. 571)
- Create and Modify a Particles Details Window (p. 679)

5.3. View and Save Simulation Statistics

You view and save simulation statistics either after you have finished processing a simulation or have completed enough of the processing that the data you want to analyze is available.

You can use the data directly on the Properties and Curve tabs to view an individual statistic for a property; or you can graph an entire data set by creating plots and histograms, or by exporting the data you want to a CSV file to view from a spreadsheet program, such as Microsoft Excel.

What would you like to do?

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

- View an Individual Statistic (p. 1098)
- Graph (Plot or Histogram) a Data Set Within Rocky (p. 1101)
- Export Data Out of Rocky (p. 1154)

See Also:

- Analyzing a Simulation (p. 931)

5.3.1. View an Individual Statistic

You choose to view an individual statistic when you want to know only a particular data point at a particular time step in the simulation.

What would you like to do?

- Learn more About Viewing an Individual Statistic (p. 1098)
- View an Individual Statistic (p. 1098)

See Also:

- View and Save Simulation Statistics (p. 1097)

About Viewing an Individual Statistic

The steps you take to view an individual statistic depend upon whether the property you are interested in has been calculated yet. Sometimes Rocky calculates other individual statistics with the statistic you have chosen if doing so will save on calculation time.

IDENTIFYING WHETHER OR NOT DATA HAS BEEN CALCULATED

You will know whether a property has been calculated or not by what operational values (Min, Max, and so on) are displayed in the panel tab, as explained below and illustrated in Figure 5.79: Property column header options in the Data Editors panel (p. 1099):

- If you see question marks <?> or <unknown> in the property row, the data has not yet been calculated.
- If you see values <numbers> in the property row, the data has already been calculated.
- If you see -1.#IND in the property row, the calculation is invalid for that particular time step. This can happen if there is no particle in the simulation at the time selected, for example.

(See also Recognize Data Placeholders (p. 116).)

ABOUT VIEWING ALREADY CALCULATED STATISTICS

For properties that are already calculated, viewing an individual statistic involves either hovering over the property and viewing the information in the tooltip, or customizing the column headers for the panel tab so that the values you want to view are visible in the UI.

About Viewing Not Yet Calculated Statistics

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

For functions that have yet to be calculated, viewing an individual statistic involves first choosing to **Compute Statistics**, which is affected by how frequently you have chosen calculations to be updated. (See also Determine the Frequency of Calculations by Pinning/Unpinning Items (p. 115).) The results of the calculation are dependent upon what kind of data item you chose, as described below:

- For items on the **Properties** tab, choosing to **Compute Statistics** results in only the current time step being calculated for the function chosen.
- For items on the **Curves** tab, choosing to **Compute Statistics** results in all currently available time steps being calculated for the property chosen. But unlike pinning an item, choosing to **Compute Statistics** will not continue to recalculate those values should settings change that affect them, nor if new time steps are added to the simulation.

COLUMN OPTIONS FOR PROPERTY AND CURVE ROWS

Use the image and table below to help you understand how you can view an individual statistic in the Rocky UI.

Figure 5.79: Property column header options in the Data Editors panel

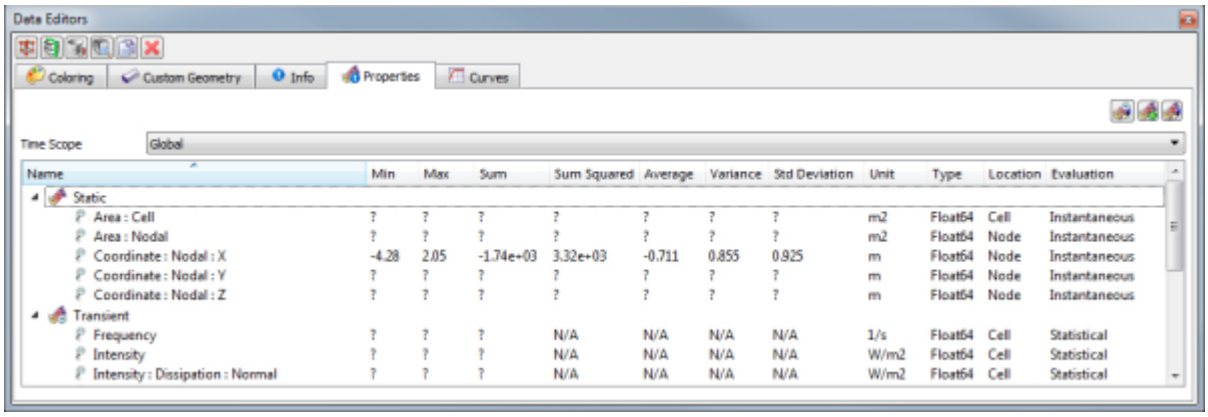


Table 1: Properties and Curves column header options

Header	Description
Name	The name of the property or curve.
Value	For Curves only, indicates the base value for the selected curve.
Min	The lowest value of all the recorded data for the property or curve.
Max	The highest value of all the recorded data for the property or curve.
Sum	The final value when of all the recorded data for the property or curve are added together.
Sum Squared	The final value when of all the recorded data for the property or curve are added together and squared.
Average	The average value of all the recorded data for the property or curve.
Variance	The variance between of all the recorded data for the property or curve.
Std. Deviation	The standard deviation between of all the recorded data for the property or curve.
Unit	The unit used by the value.

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

Header	Description
Type	For Properties only, indicates the underlying data type storing the value.
Location	For Properties only, indicates the specific part of the entity to which the property is applied. For example: • **Cell** is equal to one value per geometry triangle. • **Node** is equal to one value per triangle vertex.
Evaluation	For Properties only, defines whether the property is "Instantaneous" or "Statistical". Specifically: • **Instantaneous** properties are the values that the entity had at the moment the output file was written. These most often indicate "standard" properties that are provided in Rocky by default. • **Statistical** properties are the result of some kind of computation over the values between two consecutive output files. These most often indicate that data collection had to be turned on prior to processing the simulation, as with collision statistics. (See also About Collision Statistics for Boundaries (p. 312).)

What would you like to do?

- View an Individual Statistic (p. 1098)
- Learn more About the Property User Process (p. 1202)
- Learn more About Curves (p. 989)

See Also:

- View and Save Simulation Statistics (p. 1097)

View an Individual Statistic

1. Use the Time toolbar to select the exact point in the simulation for which you want information. (See also About the Time Toolbar (p. 1279).)
2. From the **Data** panel, select the entity containing the property for which you want to view an individual statistic. This can include a conveyor or imported geometry component, Particles, 1-Way LBM, 1-Way Fluent, Solver, or any individual User Processes linked to these entities. (See also Use Properties and Curves (p. 931).)
3. From the **Data Editors** panel, select either the **Properties** or **Curves** tab containing the property for which you want to view an individual statistic.
4. Do one or both of the following:
   - If the information has yet to be calculated (e.g., the values show question marks <?> or <unknown>), right click the property you want, and then click **Compute Statistics**. The values for the selected time step and property are displayed on the tab.

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

- If the information has already been calculated, display the information in one of the following ways:
  - In a row of the panel tab by right-clicking the header row for the property table, and then left-clicking the values you want shown/hidden from the menu displayed.
  - From a tooltip by hovering your mouse over the property.

See Also:

- About Viewing an Individual Statistic (p. 1098)

5.3.2. Graph (Plot or Histogram) a Data Set Within Rocky

In Rocky, you create plots—such as Time Plots or Cross Plots—and histograms when you want to see a graphical comparison of various data sets in the UI.

After selecting your data and displaying it in the plot or histogram window of your choice, you can then modify your graph by changing the window and data properties of your choosing.

What would you like to do?

- Learn more about Graphing (Plot or Histogram) a Data Set Within Rocky (p. 1101)
- Learn more about Changing the Appearance of a Graph (Plot or Histogram) (p. 1125)

See Also:

- View and Save Simulation Statistics (p. 1097)

Graphing (Plot or Histogram) a Data Set Within Rocky

In Rocky, there are five different types of graphs you can create with your simulation data, as described below:

- **Time Plot**: Graphs data across a time period. Also enables you to inspect a particular point in a curve and see the results in table format.
- **Multi Time Plot**: Like a Time Plot, graphs data across a time period but enables multiple plots to be compared in one window.
- **Histogram**: Provides a graphical representation of the distribution of a data set. It represents in vertical bars the data set previously tabulated and divided into uniform bins. The base of each bar represents a class and the height of each bar represents the amount or frequency with which the value of the class occurred in the data set.
- **Space Plot**: Graphs one-dimensional (line) data from the Properties tab.
- **Cross Plot**: Displays in a dot-cloud format correlations between two entity (p. 1395) properties (Curves or Properties) that share a common domain.

Creating one of these graphs produces a window within the Rocky workspace that you can then drag and drop a Property or a Curves data set into (see also Use Properties and Curves (p. 931).). From

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

there, you may use the options on the Window Editors panel to change the appearance of your graph, including settings like background color, font color/sizes, axis and legend information.

For Histograms only, you can also determine by which method you want the data weighted through the **Configure Histogram** dialog. This also enables you to change the number of bins, and whether the data is aggregated by percentage and/or by cumulative methods.

For Time Plots and Multi Time Plots only, you can also add an Annotation, which places a mark and caption in a location of your choice on the graph. (See also About Creating an Annotation on a Time Plot or Multi Time Plot (p. 1145).)

Once your graph contains the data you want and appears the way you prefer, you may then choose to save the graph as an image or export it to an external CSV file (see also About Exporting Data Out of Rocky (p. 1155).)

What would you like to do?

- Learn more About Time Plots (p. 1102)
- Learn more About Multi Time Plots (p. 1110)
- Learn more About Histograms (p. 1112)
- Learn more About Space Plots (p. 1118)
- Learn more About Cross Plots (p. 1120)
- Learn more about Changing the Appearance of a Graph (Plot or Histogram) (p. 1125)
- Graph Data within Rocky by Creating a Plot or Histogram (p. 1124)

See Also:

- Graph (Plot or Histogram) a Data Set Within Rocky (p. 1101)

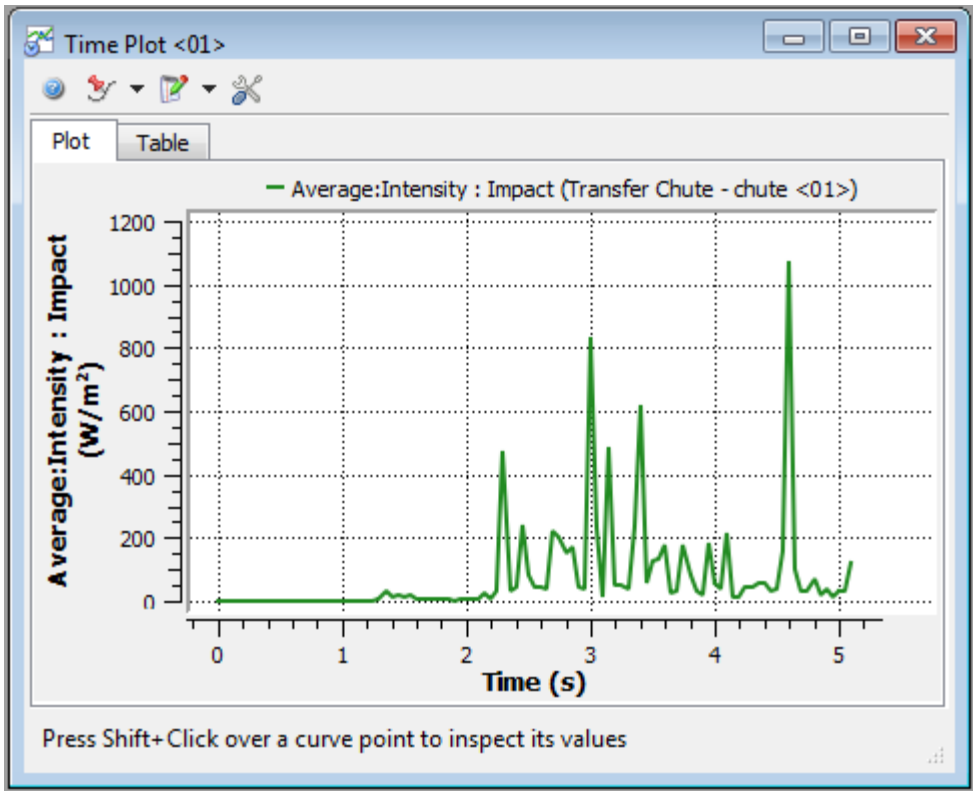
ABOUT TIME PLOTS

A Time Plot graphs data across a time period (Figure 5.80: Example Time Plot (p. 1103)). Like any other Plot in Rocky, you can inspect a particular point in a curve or zoom-in on a particular area (Figure 5.81: Inspecting a Time Plot Curve with Shift+Click (top left), Ctrl+Click (top right), and Ctrl+Click+Drag (bottom) (p. 1104)). Unlike Multi Time Plots, which accepts many sub-plots in one window, Time Plot windows accept only one plot at a time. However, Time Plots are the only graphs within Rocky with the ability to view and add custom formulas to the results in table format (Figure 5.82: Table tab results of the example Time Plot (p. 1106)).

ABOUT THE PLOT TAB

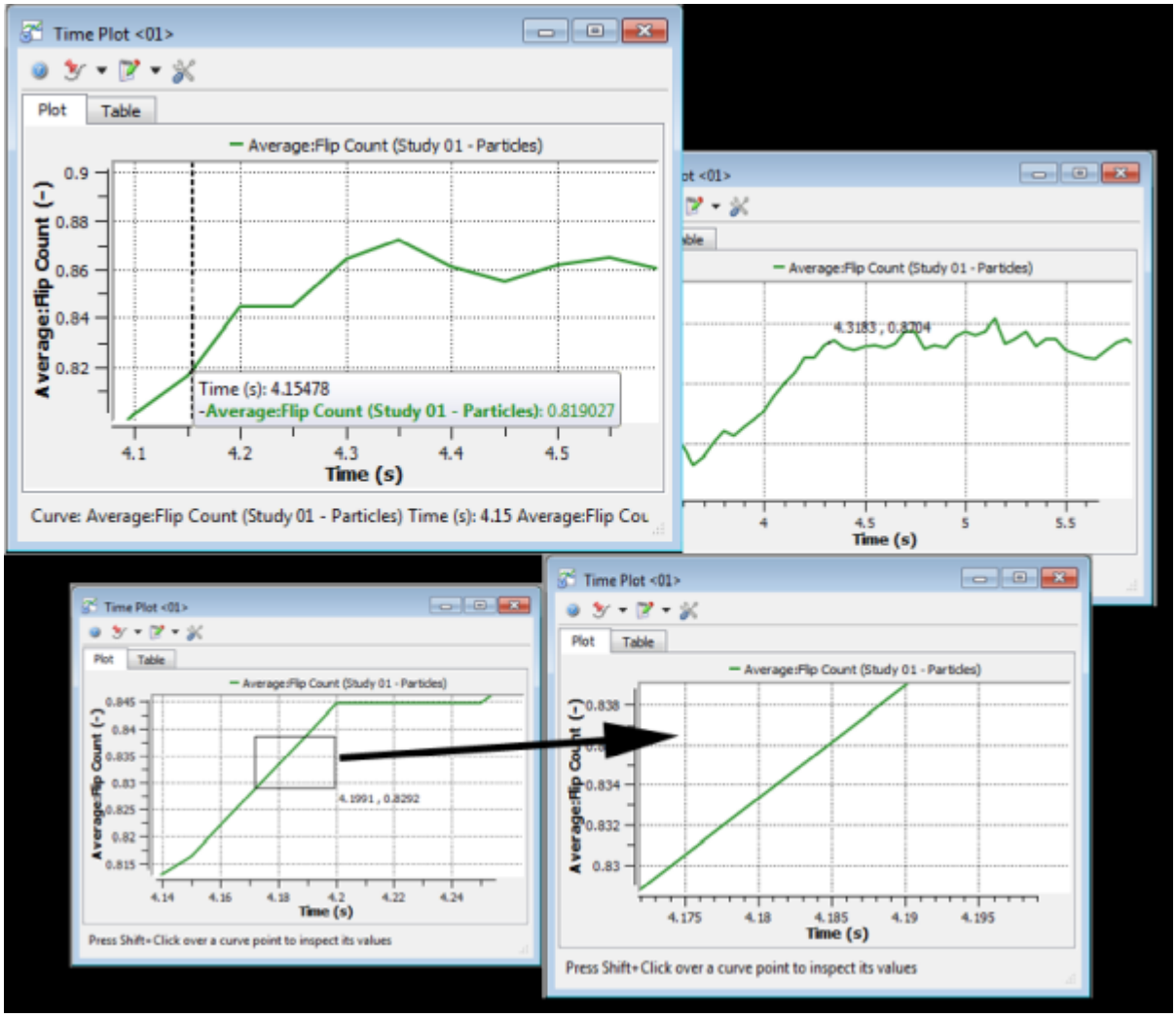
Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

Figure 5.80: Example Time Plot



Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

Figure 5.81: Inspecting a Time Plot Curve with Shift+Click (top left), Ctrl+Click (top right), and Ctrl+Click+Drag (bottom)



The Plot tab of a Time Plot window is where the graph itself is located. As with Multi Time Plots, you can add an Annotation or a Time Mark to a Time Plot. An Annotation places a mark and caption in a location of your choice on the graph (see also About Creating an Annotation on a Time Plot or Multi Time Plot (p. 1145)) while a Time Mark enables you to identify with a vertical line the location of the current time step.

Tip:

To see walk-through examples of Time Plots being used to analyze simulation results, refer to the following Tutorials:

- Tutorial - Transfer Chute in the *Rocky Tutorial Guide*
- Tutorial - High Pressure Grinding Roll (HPGR) in the *Rocky Tutorial Guide*
- Tutorial - Static Structural in the *Rocky Tutorial Guide*

ABOUT THE TABLE TAB

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

The Table tab for a Time Plot (Figure 5.82: Table tab results of the example Time Plot (p. 1106)) enables you to view the data in tabular format and also add and edit custom formulas using variables and expressions (see also Defining Your Variables (p. 243)) that will themselves be calculated and added to the graph.

See the images and tables below to understand how you can use the options on the Table tab for a Time Plot window to change the data displayed in your graph.

Tip:

To see walk-through examples of the Table tab being used to analyze simulation results, refer to the following Tutorials:

- Tutorial - Vibrating Screen in the *Rocky Tutorial Guide*
- Tutorial - Drop Weight Test in the *Rocky Tutorial Guide*
- Tutorial - Conical Dryer in the *Rocky Tutorial Guide*

TABLE TAB PARAMETER DEFINITIONS

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

Figure 5.82: Table tab results of the example Time Plot

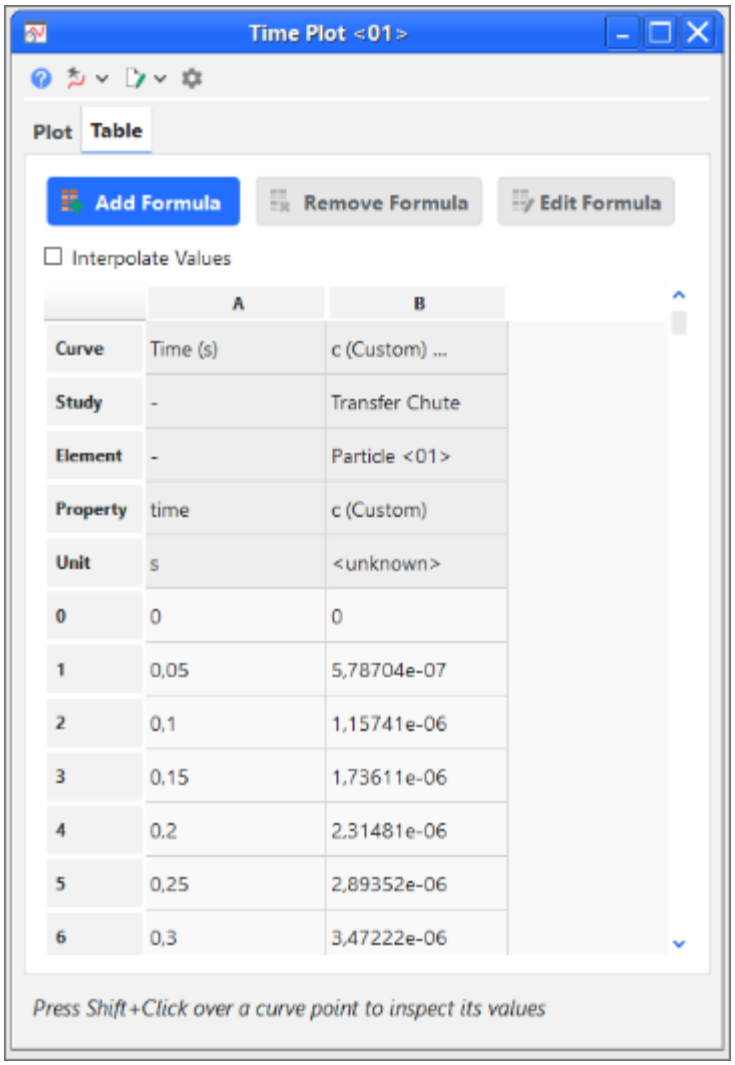


Table 1: Table tab options for a Time Plot window

Setting	Description	Range
Interpolate Values	When enabled, this will create the non-existing values so that when there are multiple curves plotted, all curves have values in all exported time steps.	Turns on or off

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

Figure 5.83: Add Curve Expression dialog

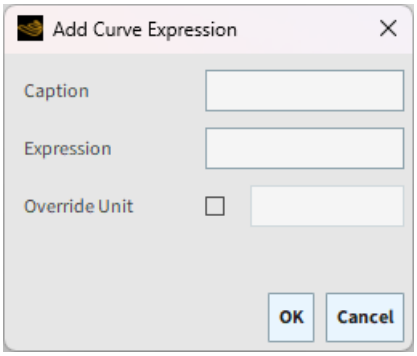


Table 2: Add Curve Expression dialog options

Setting	Description	Range
Caption	The name you want to give the new curve. This will be displayed in the Table as a new column and in the Time Plot legend. **Note:** Unlike curves you drag and drop into the Time Plot window, you cannot edit curves added from the Table tab in the Window Editors panel. Rather, you must only use the options in the Edit Expression dialog.	No limit
Expression	The mathematical expression for the curve you want to add to the Time Plot.	
Override Unit	Allows you to set a custom unit for the curve you are adding. **Note:** If the unit you set is not recognized by Rocky, you might see a "Unit not recognized by the internal system" warning. However, this will not prevent you from creating the curve.	

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

Figure 5.84: Edit Expression dialog

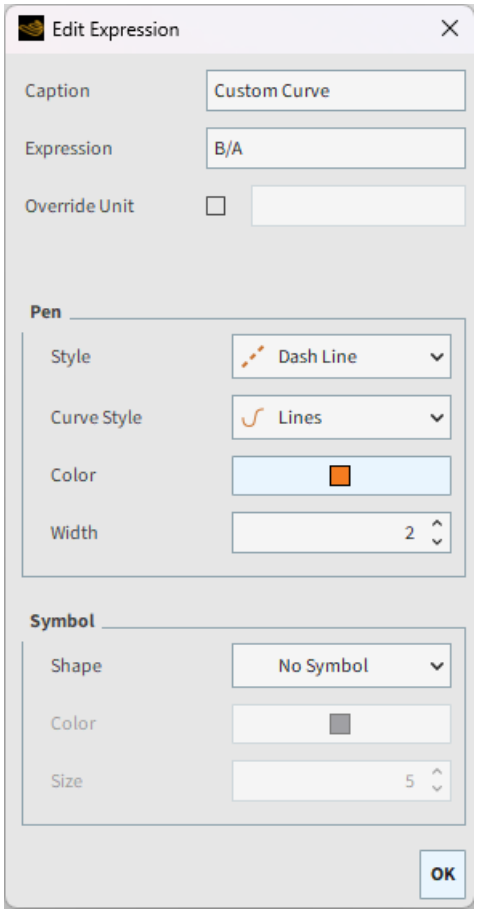


Table 3: Edit Expression dialog options

Setting	Description	Range
Caption	Enables you to change the name of the curve you are editing. This will be displayed in the Table as a column header and will also be listed in the Time Plot legend and axis titles. **Note:** Unlike curves you drag and drop into the Time Plot window, you cannot edit curves added from the Table tab in the Window Editors panel. Rather, you must only use the options in the Edit Expression dialog.	No limit
Expression	Enables you to change the mathematical expression for the curve you are editing.	

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

Setting	Description	Range
Override Unit	Allows you to set a custom unit for the curve you are adding. **Note:** If the unit you set is not recognized by Rocky, you might see a "Unit not recognized by the internal system" warning. However, this will not prevent you from creating the curve.	
Pen		
Style	Enables you to set what kind of line style you want represented on the plot for the custom curve. **Tip:** If you want the curve represented on the plot by symbols instead of lines, select **No Line** and then choose what you want under **Symbol**.	No Line; Solid Line; Dash Line; Dot Line; Dash Dot Line; Dash Dot Dot Line
Curve Style	When you select a **Style** other than **No Line**, this enables you to set how the curve is represented in the plot. What you set here will be drawn in the **Style** indicated.	Lines; Sticks; Steps
Color	When you select a **Style** other than **No Line**, this enables you to change the color of the plot line for the custom curve.	Options limited by the **Select Color** dialog
Width	When you select a **Style** other than **No Line**, this enables you to change the width of the plot line for the custom curve.	Whole numbers greater than or equal to one
Symbol		
Shape	Enables you to add to the plot a symbol marking the custom curve data point for each recorded time step. (For example, every 0.05 s.) **Tip:** If you want the curve represented on the plot by lines instead of symbols, select **No Symbol** and then choose what you want under **Pen**.	Ellipse; Rectangle; Diamond; Triangle; Down Triangle; No Symbol
Color	Enables you to change the color of the symbol used to mark the data points of the custom curve.	Options limited by the **Select Color** dialog
Size	Enables you to change the size of the symbol used to mark the data points of the custom curve.	Whole, positive values

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

What would you like to do?

- Graph Data within Rocky by Creating a Plot or Histogram (p. 1124)
- Learn more About Changing the Appearance of a Graph (Plot or Histogram) (p. 1126)
- Add and Edit Custom Table Formulas in a Time Plot (p. 1152)

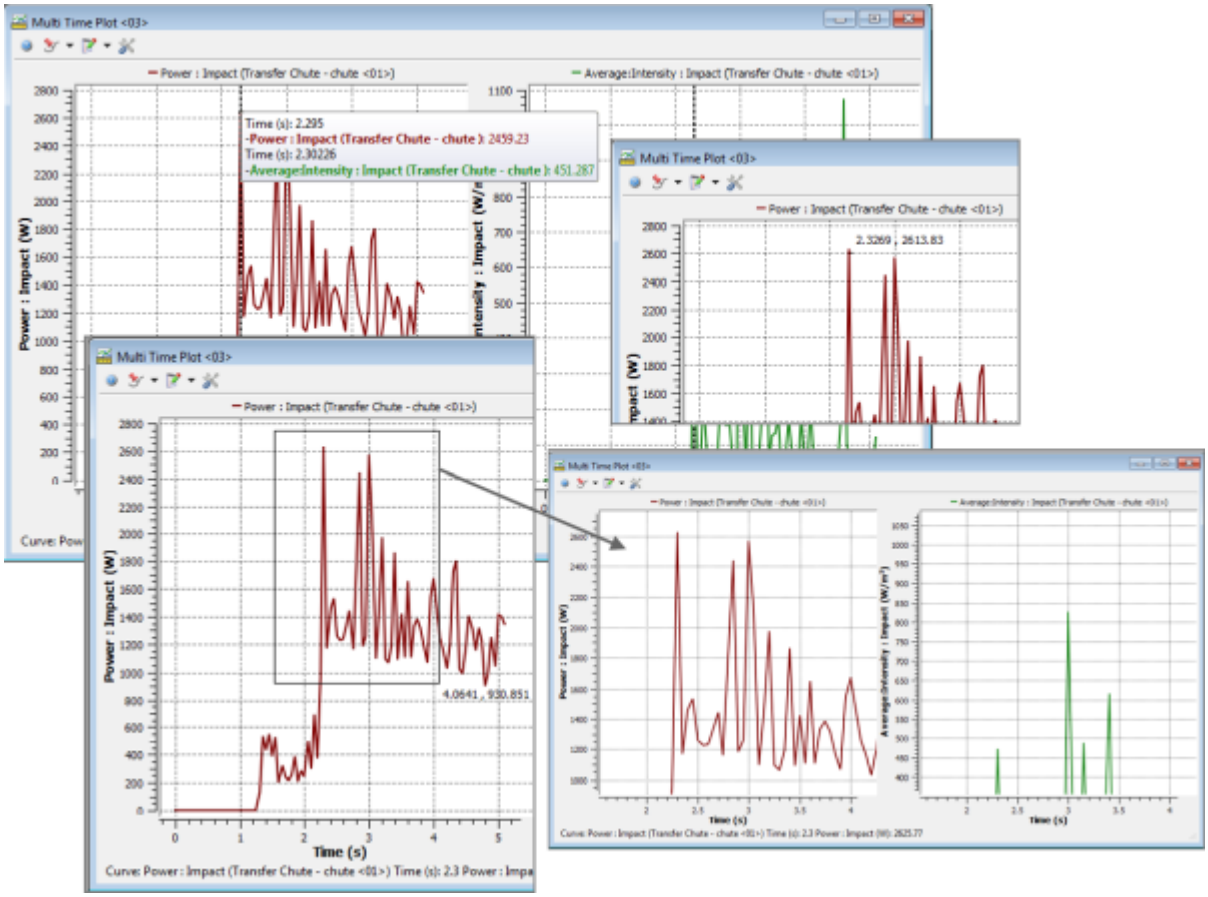
See Also:

- About Graphing (Plot or Histogram) a Data Set Within Rocky (p. 1101)
- Graph (Plot or Histogram) a Data Set Within Rocky (p. 1101)

ABOUT MULTI TIME PLOTS

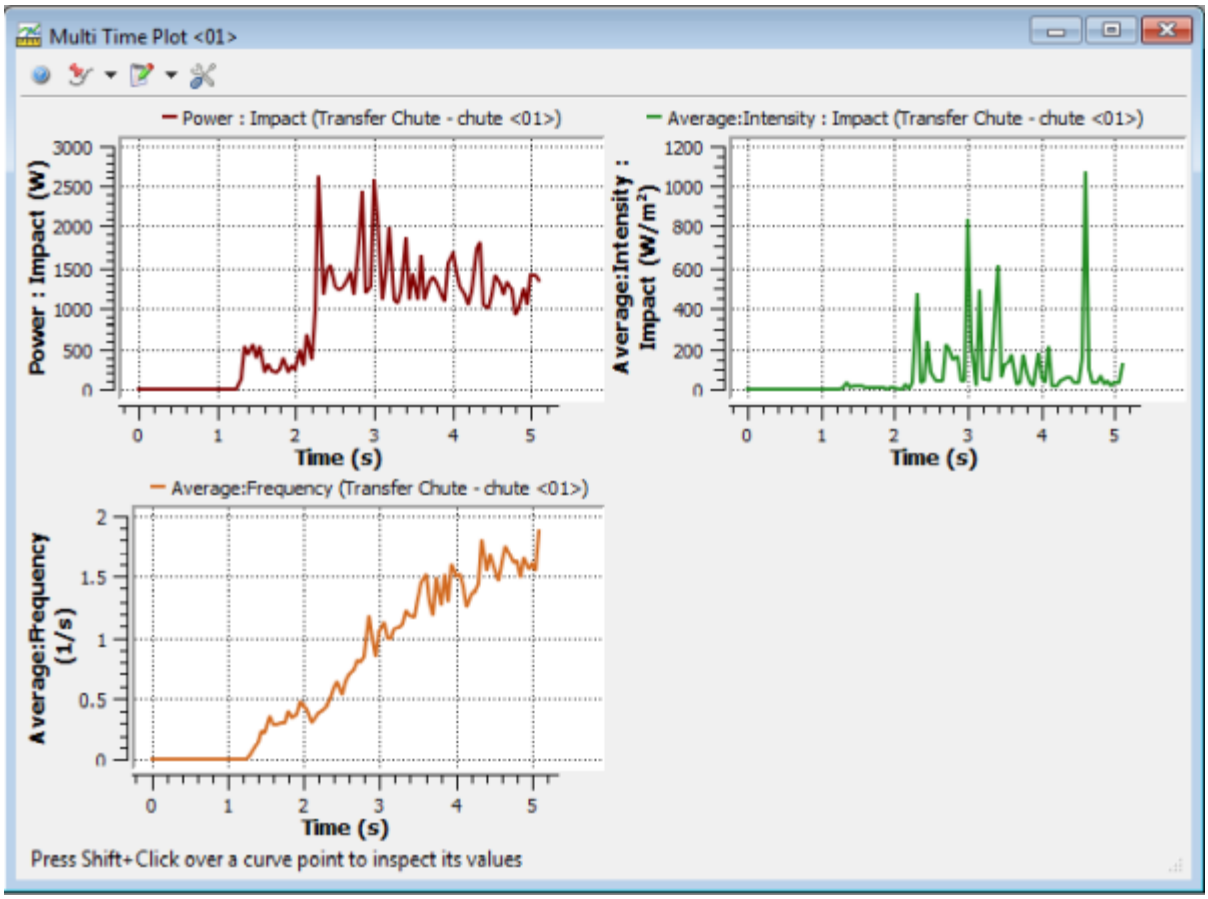
A Multi Time Plot is a type of graph that like a Time Plot, graphs data across a time period while also enabling you to inspect a particular point in a curve or zoom-in on a particular area (as shown below). But unlike a Time Plot, a Multi Time Plot enables multiple plots, called sub-plots, to be compared in one graphical window:

Figure 5.85: Inspecting a Multi-Time Plot Curve with Shift+Click (top left), Ctrl+Click (top right), and Ctrl+Click+Drag (bottom)



Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

Figure 5.86: Example Multi Time Plot showing three sub-plots



You add additional sub-plots to an existing Multi Time Plot window by selecting the property (Property or Curves) that you want and then holding the Ctrl key while you drop it in the window. This will create another plot within the same window. If you do not hold the Ctrl key before dropping the property, instead of being added as a separate sub-plot, the property will be added to the same sub-plot over which you released your mouse over. Tip:While dragging, use the tooltip and hover text to help you determine how and where to drop your data for different effects.

All sub-plots within the same window share the same display settings and can be treated as one image for export (see also Save an Image of the 3D View, Plot, or Histogram (p. 858)).

Like with Time Plots, you can also add an Annotation or a Time Mark to a Multi Time Plot. An Annotation places a mark and caption in a location of your choice on the graph (see also About Creating an Annotation on a Time Plot or Multi Time Plot (p. 1145)) while a Time Mark enables you to identify with a vertical line the location of the current time step.

In addition to the standard settings available to all graphs, you can also control how these sub-plots appear in the window by using the Layout tab on the Window Editors panel.

Tip:

To see walk-through examples of simulations that make use of Multi Time Plots, refer to the following Tutorials:

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

- Tutorial - Transfer Chute in the *Rocky Tutorial Guide*
- Tutorial - Drop Weight Test in the *Rocky Tutorial Guide*

What would you like to do?

- Graph Data within Rocky by Creating a Plot or Histogram (p. 1124)
- Learn more About Changing the Appearance of a Graph (Plot or Histogram) (p. 1126)

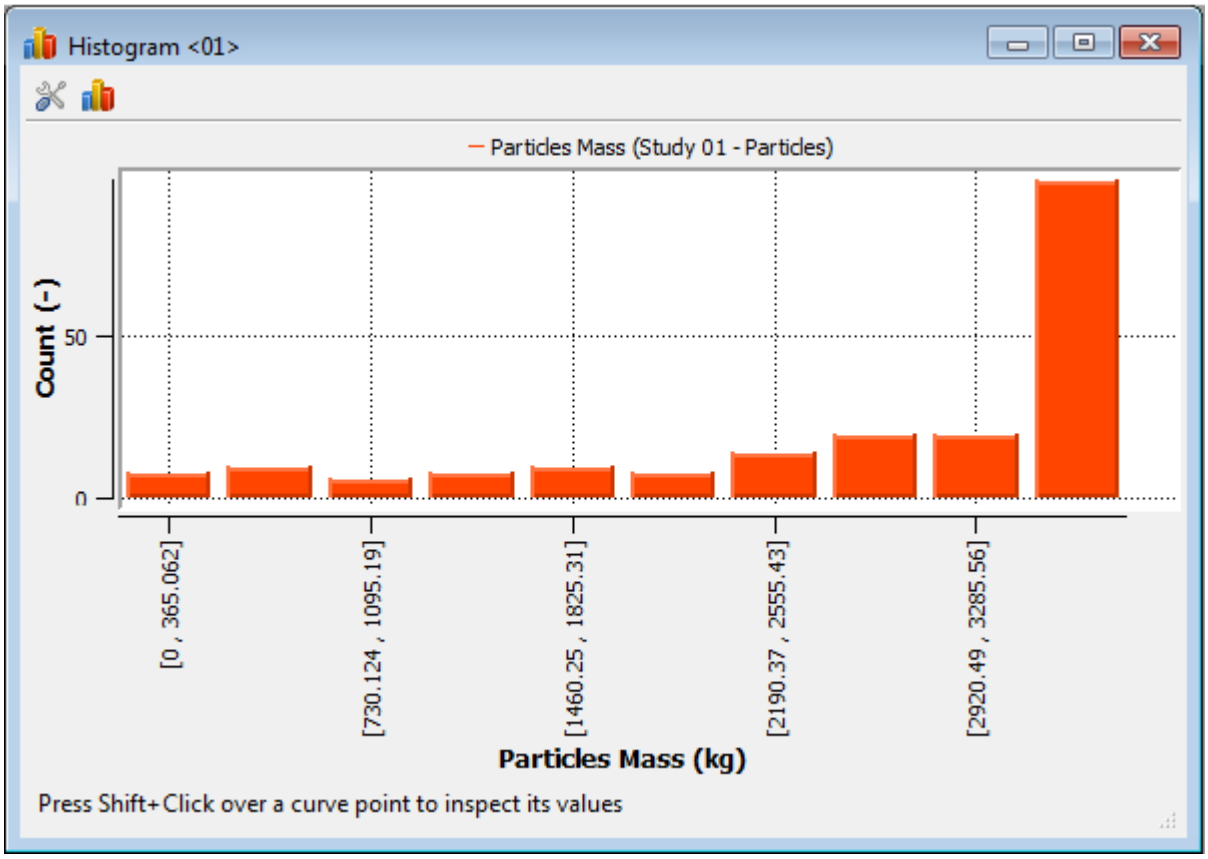
See Also:

- About Graphing (Plot or Histogram) a Data Set Within Rocky (p. 1101)
- Graph (Plot or Histogram) a Data Set Within Rocky (p. 1101)

ABOUT HISTOGRAMS

A Histogram is a type of graph that represents in vertical bars the data set previously tabulated and divided into uniform bins. The base of each bar represents a class and the height of each bar represents the amount or frequency with which the value of the class occurred in the data set:

Figure 5.87: Example Histogram



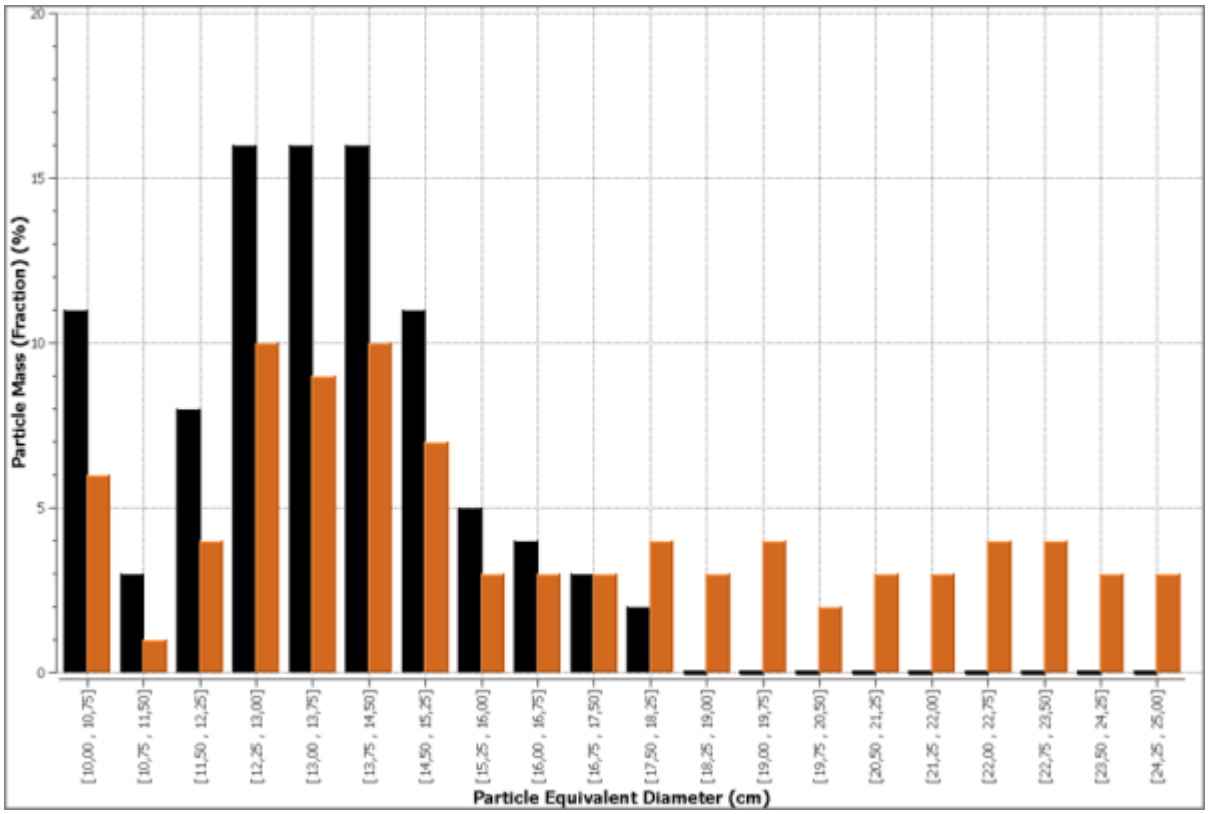
CONFIGURING HISTOGRAMS

Histograms work by aggregating data into separate bins for analysis. One of the benefits of this type of graph is that you can weigh and compare different statistics for different results. For example, you

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

can compare the results of one statistic weighted by another statistic at two different times in the simulation:

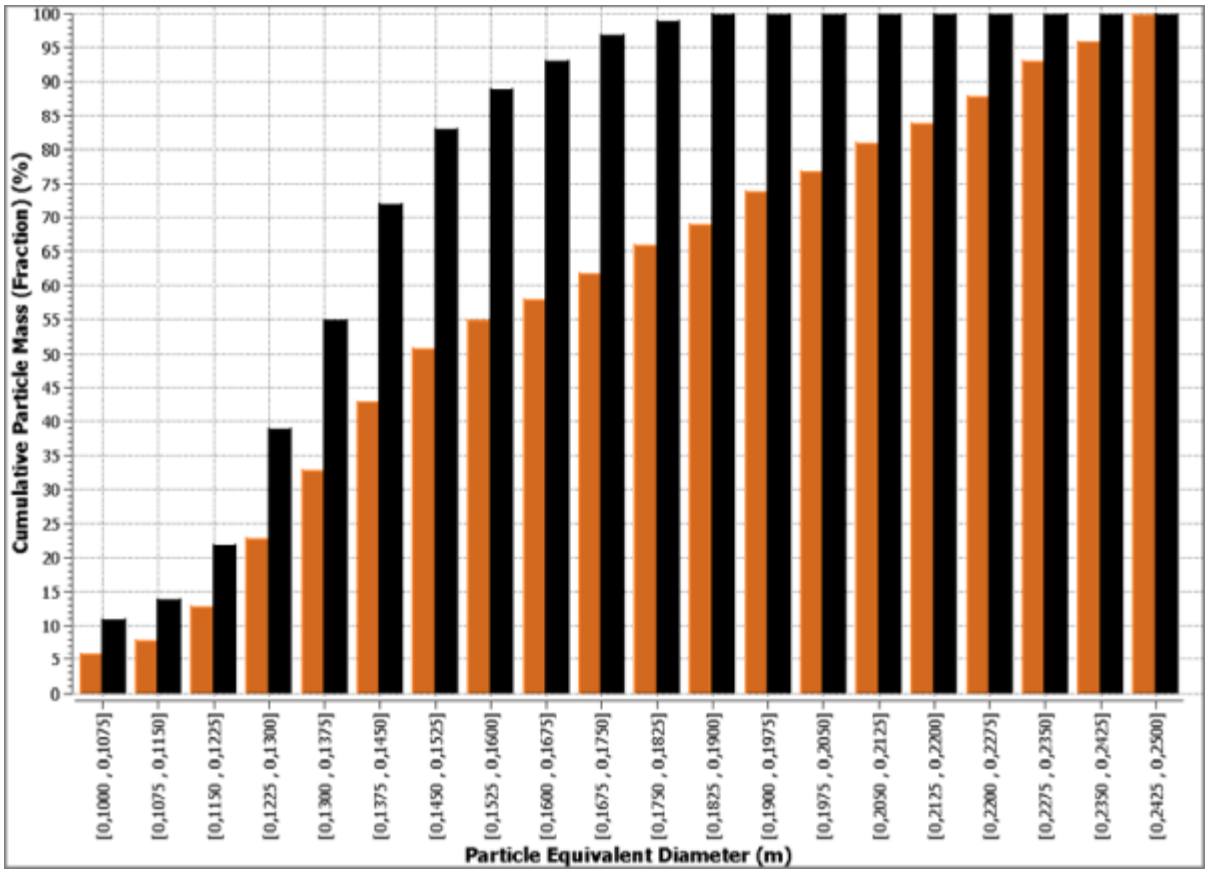
Figure 5.88: Example Histogram showing a comparison of weighted statistics



Histograms enable you to have even more control over your analysis by enabling you to plot your data in a percentage and/or cumulative manner:

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

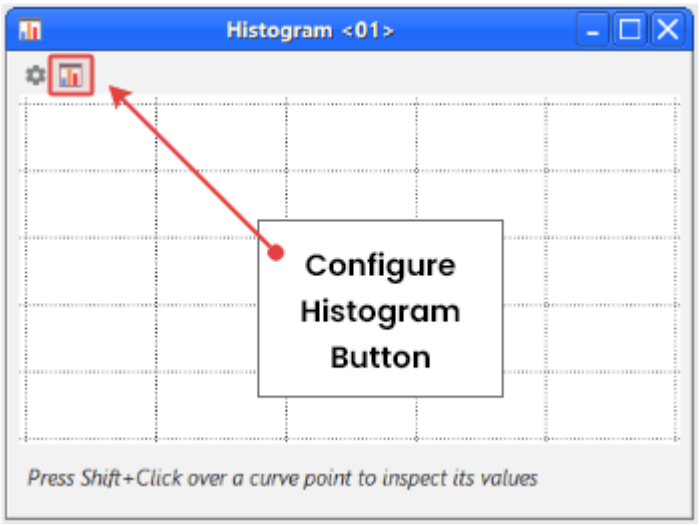
Figure 5.89: Example Histogram showing a cumulative percentage of weighted statistics



These settings are made through a separate **Configure Histogram** dialog that you access from the top left corner of the Histogram window:

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

Figure 5.90: Location of the Configure histogram button



Tip:

To see walk-through examples of simulations making use of Histograms, refer to the following Tutorials:

- Tutorial - Vibrating Screen in the *Rocky Tutorial Guide*
- Tutorial - Drop Weight Test in the *Rocky Tutorial Guide*
- Tutorial - High Pressure Grinding Roll (HPGR) in the *Rocky Tutorial Guide*

CONFIGURE HISTOGRAM PARAMETER DEFINITIONS

See the image and table below to help you understand how to configure a Histogram.

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

Figure 5.91: Configure Histogram dialog showing one statistic weighted by another

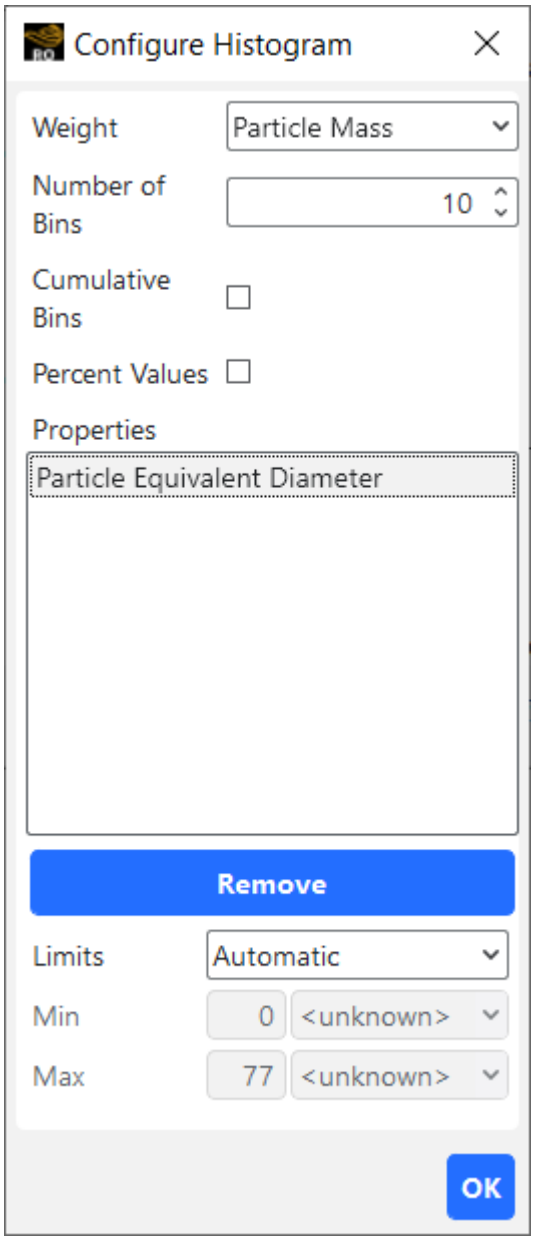


Table 1: Configure Histogram options

Setting	Description	Range
Weight	The method by which you want the selected property(s) weighted for comparison. The weight you choose affects all properties for the selected Histogram. The weight options can include the following:	Properties (for both geometries and

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

Setting	Description	Range
	• **Properties**: If you chose to display a Property in your Histogram, then Properties for both geometries and particles will be listed here. (See also About Properties (p. 933)). **Important:** Though you are able to select any Property you want from the **Weight** list, only those Properties that are applicable to the **Properties** you have selected will be shown in the Histogram. For example, if you want to display a Particles Property like Particle Volume, you cannot weigh it by a geometries Property like Area. Doing so will remove the data from your Histogram window until you select an applicable option. • **Count**: This is the default option for both Properties and Curves.	particles); Count
Number of Bins	Enables you to choose the number of bins or bars the data is aggregated into.	Whole numbers greater than 1
Cumulative Bins	When selected, plots in ascending order the relative frequency of each class (bin) added to all classes (bins) with lower values. A useful method for showing a range of values, this is a mapping that counts the cumulative number of observations in all of the classes (bins) up to the specified bin.	Turns on or off
Percent Values	When selected, provides the weighted value by percentage.	Turns on or off
Properties	Displays the name of the Property(s) and/or Curve(s) that you have added to the Histogram. (See also Graph Data within Rocky by Creating a Plot or Histogram (p. 1124)). These are the items that will be affected by the **Weight** you select.	Automatic display
Limits	Enables the values of the selected item in the **Properties** list to be changed. The options are as follows: • **Automatic**, which displays the limits as calculated by Rocky for the current time step. • **User Defined**, which enables you to enter a custom **Min** and **Max** value.	Automatic; User Defined
Min	When **Limits** is set to **Automatic**, this displays the minimum value for the selected property as calculated for the current time step. (See also About the Time Toolbar (p. 1279)). When **Limits** is set to **User Defined**, this enables you to set a custom minimum value	No limit

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

Setting	Description	Range
	for the selected property that you want displayed in the Histogram.	
Max	When **Limits** is set to **Automatic**, this displays the maximum value for the selected property as calculated for the current time step. (See also About the Time Toolbar (p. 1279)). When **Limits** is set to **User Defined**, this enables you to set a custom maximum value for the selected property that you want displayed in the Histogram.	No limit

What would you like to do?

- Configure how Data is Displayed in a Histogram (p. 1151)

See Also:

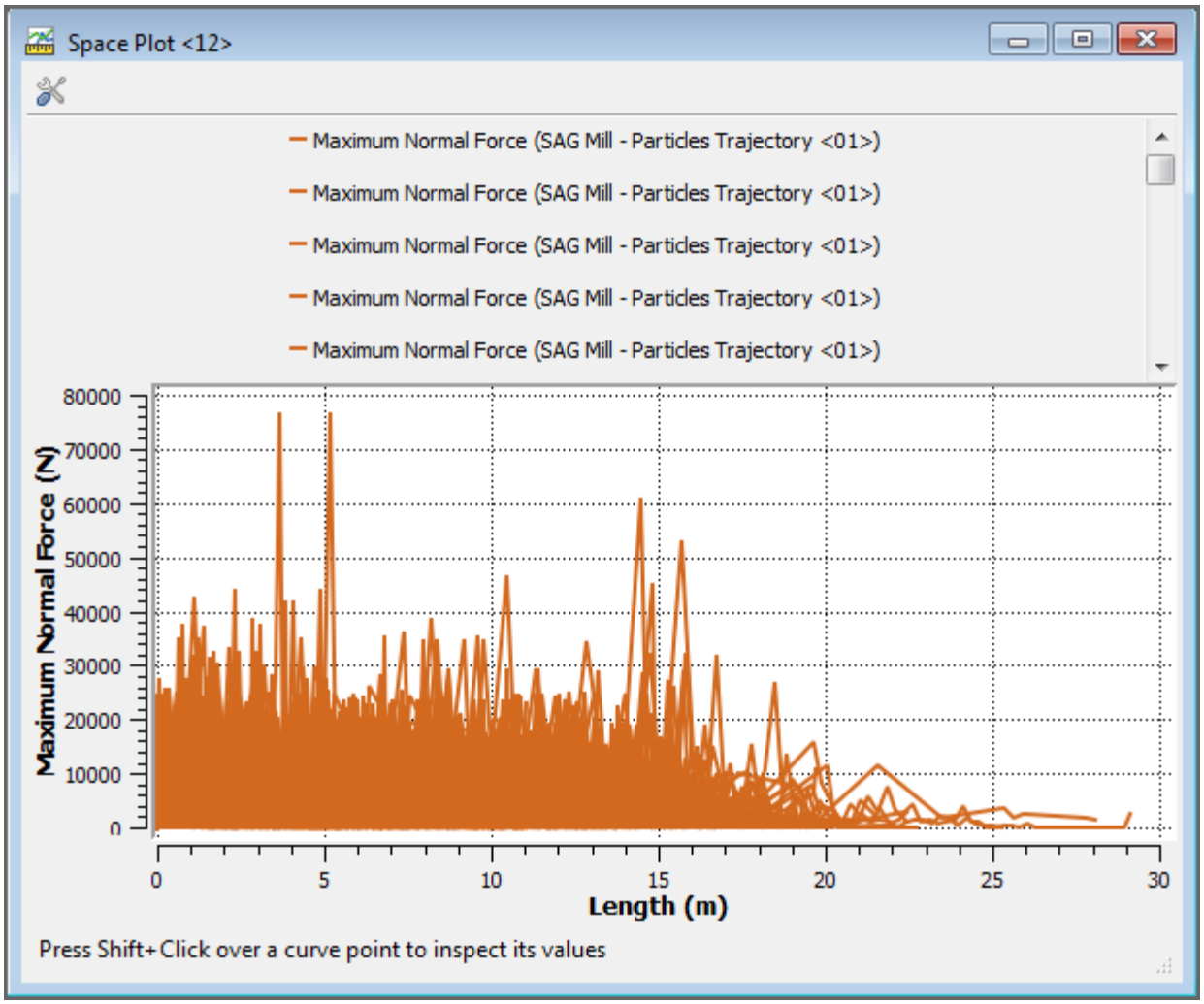
- About Graphing (Plot or Histogram) a Data Set Within Rocky (p. 1101)
- About Changing the Appearance of a Graph (Plot or Histogram) (p. 1126)
- Graph (Plot or Histogram) a Data Set Within Rocky (p. 1101)

ABOUT SPACE PLOTS

Space Plots are graphs that plot one-dimensional information from the Properties tab (Figure 5.92: Example of a Particle Trajectory property being graphed in a Space Plot (p. 1119)).

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

Figure 5.92: Example of a Particle Trajectory property being graphed in a Space Plot



While most data in Rocky does not have the one-dimensional criteria required for this kind of graphing, Space Plots can still be useful in certain specific conditions such as the following:

- Particle Trajectory Properties (see also About the Particles Trajectory User Process (p. 1209))
- Cut Plane Properties (see also About the Plane User Process (p. 1195))
- Worn Geometry Properties (see also View Surface Wear Modification on an Imported Geometry (p. 379))

What would you like to do?

- Graph Data within Rocky by Creating a Plot or Histogram (p. 1124)
- Learn more About Changing the Appearance of a Graph (Plot or Histogram) (p. 1126)

See Also:

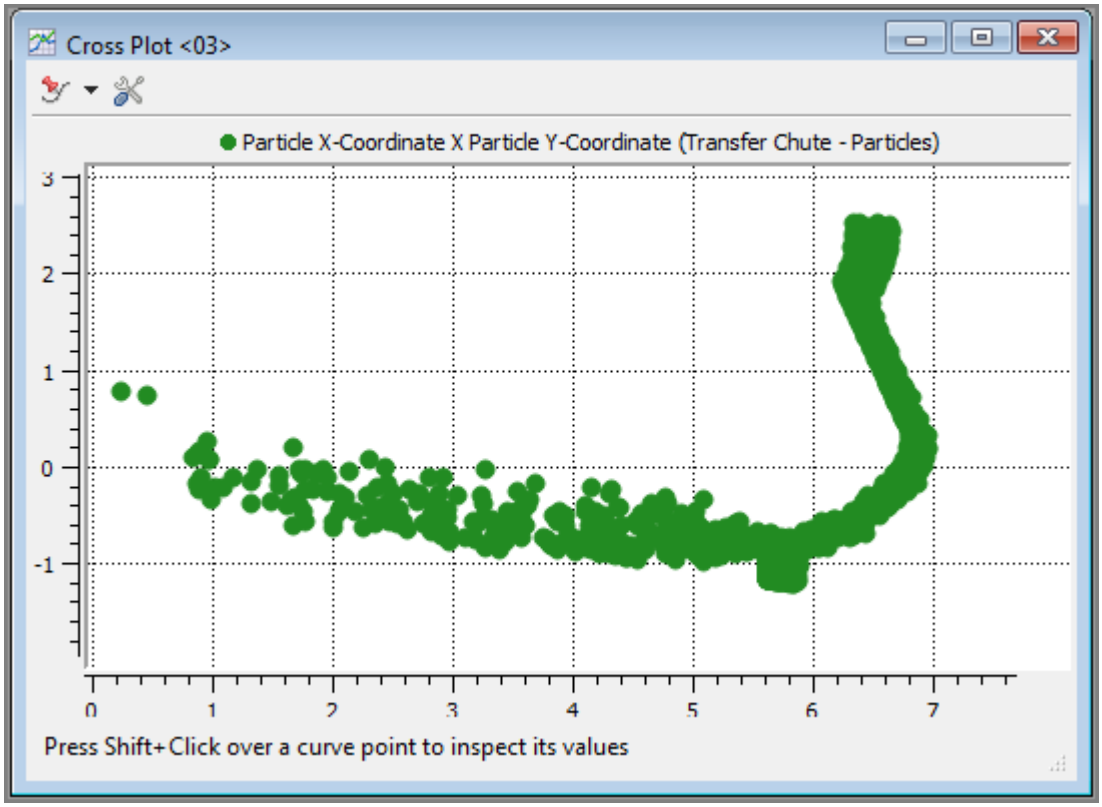
- About Graphing (Plot or Histogram) a Data Set Within Rocky (p. 1101)
- About Properties (p. 933)

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

ABOUT CROSS PLOTS

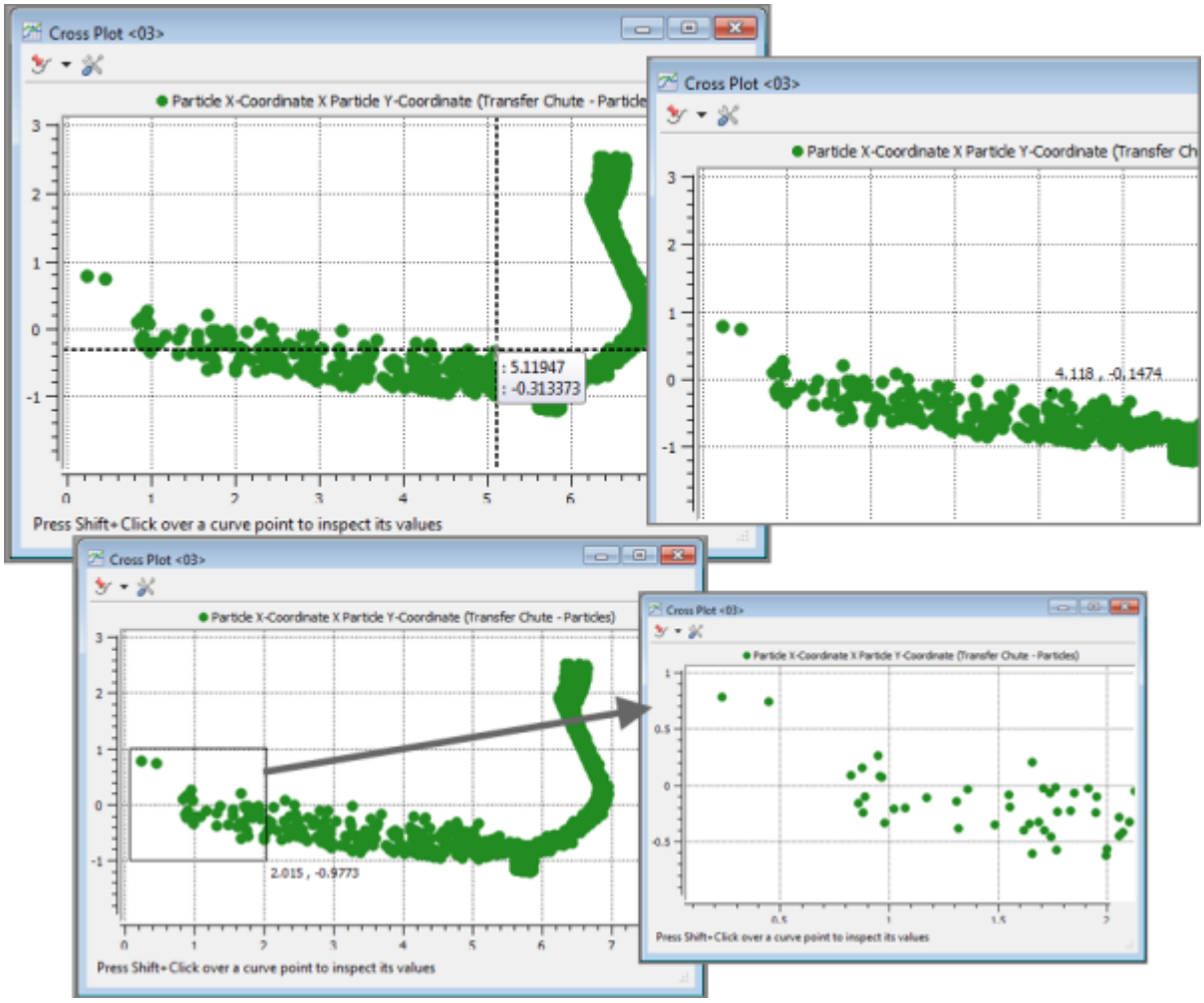
A Cross Plot is a type of graph that displays in a dot-cloud format correlations between two entity (p. 1395) properties (Curves or Properties) that share a common domain (Figure 5.93: Example Cross Plot (p. 1120)). Like other Plots, you can also inspect a particular point or zoom-in on a particular area (Figure 5.94: Inspecting a Cross Plot Dot-Cloud with Shift+Click (top left), Ctrl+Click (top right), and Ctrl+Click+Drag (bottom) (p. 1121)).

Figure 5.93: Example Cross Plot



Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

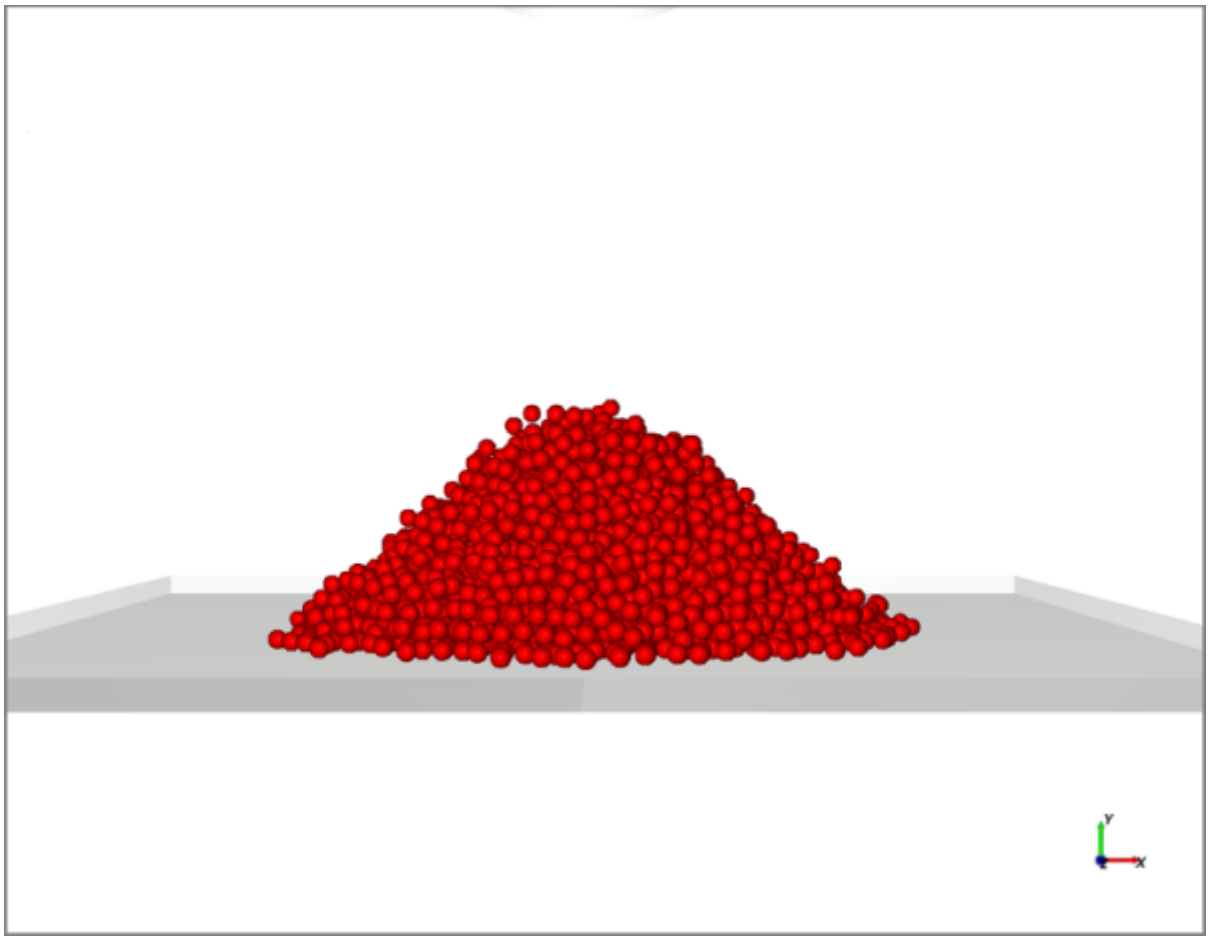
Figure 5.94: Inspecting a Cross Plot Dot-Cloud with Shift+Click (top left), Ctrl+Click (top right), and Ctrl+Click+Drag (bottom)



One example of how you can use a Cross Plot is to measure the angle of repose of your calibrated material to verify that the simulated particles match real-world results. After creating a sample pile of your particles (Figure 5.95: Simulated particle pile to be measured (p. 1122)), you can then create a Cross Plot of the Particle Y-Coordinate position (image) crossed with the Particle X-Coordinate position (domain) (Figure 5.96: Cross Plot of the above particle pile showing the Particle X-Coordinate position crossed with the Particle Y-Coordinate position (p. 1123)). Using this Cross Plot, you can then measure the angle of repose.

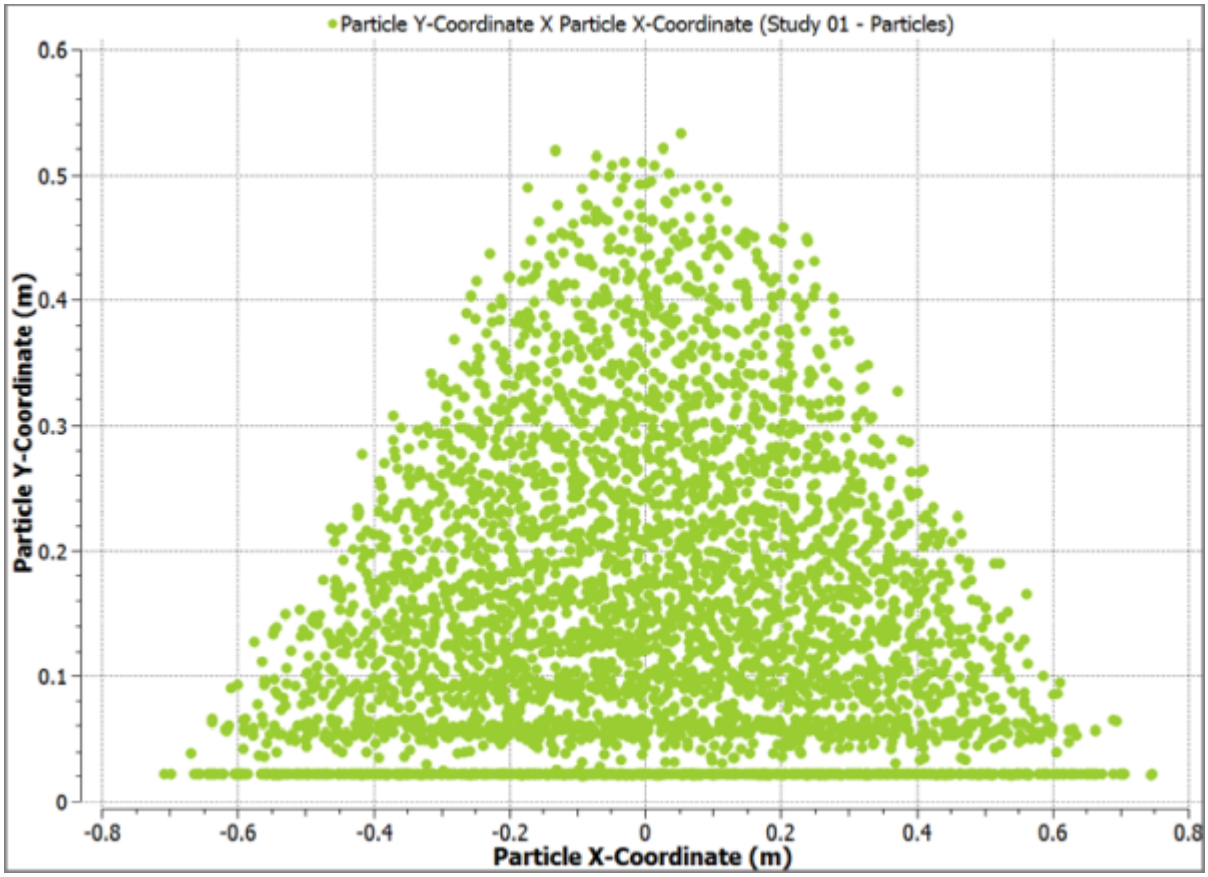
Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

Figure 5.95: Simulated particle pile to be measured



Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information
of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

Figure 5.96: Cross Plot of the above particle pile showing the Particle X-Coordinate position crossed with the Particle Y-Coordinate position



The two restrictions for creating a Cross Plot are as follows:

1. The Curves to be crossed must share a common property. For example, given an entity (Particles) that has two Curves [(Pressure X Time) and (Temperature X Time)], there must be a property common to both Curves (Time) in order to generate the desired Cross Plot (Pressure X Temperature).

2. You may only cross Curves from the same entity. For example, you may not cross a Curve from one type of entity (Particles) with another Curve from a different entity (Feed Conveyor). The Curves must come from the same entity.

Like other graphs you create in Rocky, you can also use the Window Editors panel to change how the graph appears.

Tip:

To see walk-through examples of using a Cross Plot, review the following Tutorials:

- Tutorial - Static Angle of Repose Test in the *Rocky Tutorial Guide*
- Tutorial - SAG Mill in the *Rocky Tutorial Guide*

What would you like to do?

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

- Graph Data within Rocky by Creating a Plot or Histogram (p. 1124)
- Learn more About Changing the Appearance of a Graph (Plot or Histogram) (p. 1126)

See Also:

- About Graphing (Plot or Histogram) a Data Set Within Rocky (p. 1101)
- Graph (Plot or Histogram) a Data Set Within Rocky (p. 1101)

GRAPH DATA WITHIN ROCKY BY CREATING A PLOT OR HISTOGRAM

1. Open the simulation containing the results you want to graph.
2. From the **Windows** menu, select the plot (p. 1393) or histogram that you want. (See also About Graphing (Plot or Histogram) a Data Set Within Rocky (p. 1101)) An empty window appears on the Workspace (p. 1396).
3. From the **Data** panel, select the entity (p. 1395) containing the data that you want to graph (Particles, Solver, an individual geometry component, CFD Coupling component, or an individual User Process).
4. From the **Data Editors** panel, select either the **Properties** or **Curves** tab, locate and select the function(s) you want to graph, and then click and drag the function(s) to the empty plot or histogram window you created. (See also Use Properties and Curves (p. 931)). The data you selected appears in the window.

Tips:

- Add additional data to your existing graph by repeating step 4.
- Add additional sub-plots to a Multi Time Plot by repeating step 4 but before dropping the data in the window, press and hold the Crtl key. (See also About Multi Time Plots (p. 1110).)
- Use the options on the **Window Editors** panel to change the display of your graph to your liking. (See also About Changing the Appearance of a Graph (Plot or Histogram (p. 1126).)
- View data in tabular format, add and edit custom curves for a Time Plot through the Table tab. (See also Add and Edit Custom Table Formulas in a Time Plot (p. 1152).)
- Shift-click a point along any curve to inspect the data, which will both provide a tooltip with information as well as posting it at the bottom of the window. You can also press and hold the Shift key while left-clicking and dragging your mouse along a curve to inspect multiple points.
- For Plots only (not Histograms), Ctrl-click anywhere within the plot area to see data for that point. You can also press and hold the Ctrl key while left-clicking and dragging your mouse to select an area upon which to zoom-in; after releasing the mouse, the plot will be re-drawn to the area you selected.
- Lock the Curve selection being displayed to any Plot and then use the Data panel to change the data being shown. (See also Lock a Curve Selection to a Plot to Quickly Change the Data Shown (p. 120).)
- To duplicate a Plot or Histogram, navigate to the **Windows** menu, then right-click the plot that you want and select Duplicate.

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

See Also:

- About Graphing (Plot or Histogram) a Data Set Within Rocky (p. 1101)
- Graph (Plot or Histogram) a Data Set Within Rocky (p. 1101)

Changing the Appearance of a Graph (Plot or Histogram)

Once you create your plot or histogram and add data to it, you can then modify your graph by changing the window properties and/or the data properties as described below:

- Visualization properties, like magnification, which are done through the mouse, keyboard, or toolbar.
- Window display properties, which include settings like background color, font color/sizes, axes and legend information. These are things that affect the appearance of your graph and are accomplished using the Window Editors panel. For Time Plots or Multi Time Plots, this is also where you will add Annotations, which enables you to define a specific data mark and caption on your graph.
- Data properties, which include what Properties or Curves you want to plot, and for Histograms, by what method you want the data weighted. These changes are made using Data and Data Editors panel, and for Histograms, the **Configure Histogram** dialog.

What would you like to do?

- Learn more About Using the Mouse and Keyboard to Change the View (p. 1043)
- Learn more about Using the Window Editors Panel to Change a Graph (Plot or Histogram) Window (p. 1126)
- Learn more about Using the Coloring Tab to Change a Graph (Plot or Histogram) Window (p. 1152)

See Also:

- Graph (Plot or Histogram) a Data Set Within Rocky (p. 1101)
- View and Save Simulation Statistics (p. 1097)

USE THE MOUSE, KEYBOARD, OR TOOLBAR TO CHANGE A PLOT

Note:

The movements described below do not apply to histogram windows.

---

1. Select the plot window (p. 1396) you want to change.
2. Do one or more of the following:
   - Use the left or center wheel buttons on the mouse to pan or zoom respectively.

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

- Press R on the keyboard or click the **Fit** button on the Camera visualization toolbar to automatically adjust the window view to the available data.

Tip:

Additional keyboard shortcuts are listed in the Preferences dialog. (See also Keyboard Accessibility and Shortcuts (p. 128)).

See Also:

- About Using the Mouse and Keyboard to Change the View (p. 1043)
- Changing the Appearance of a Graph (Plot or Histogram) (p. 1125)

USING THE WINDOW EDITORS PANEL TO CHANGE A GRAPH (PLOT OR HISTOGRAM) WINDOW

From the Window Editors panel, you can modify display settings for the selected plot or histogram window. Changes can include the background color, font color/sizes, and window axes displays. You can also synchronize the time steps displayed when two or more graphing windows are open.

What would you like to do?

- Learn more About Changing the Appearance of a Graph (Plot or Histogram) (p. 1126)
- Learn more About Creating an Annotation on a Time Plot or Multi Time Plot (p. 1145)
- Use the Window Editors Panel to Change the Appearance of a Graph (Plot or Histogram) (p. 1149)
- Create an Annotation on a Time Plot or Multi Time Plot (p. 1149)
- Configure how Data is Displayed in a Histogram (p. 1151)
- Add and Edit Custom Table Formulas in a Time Plot (p. 1152)

See Also:

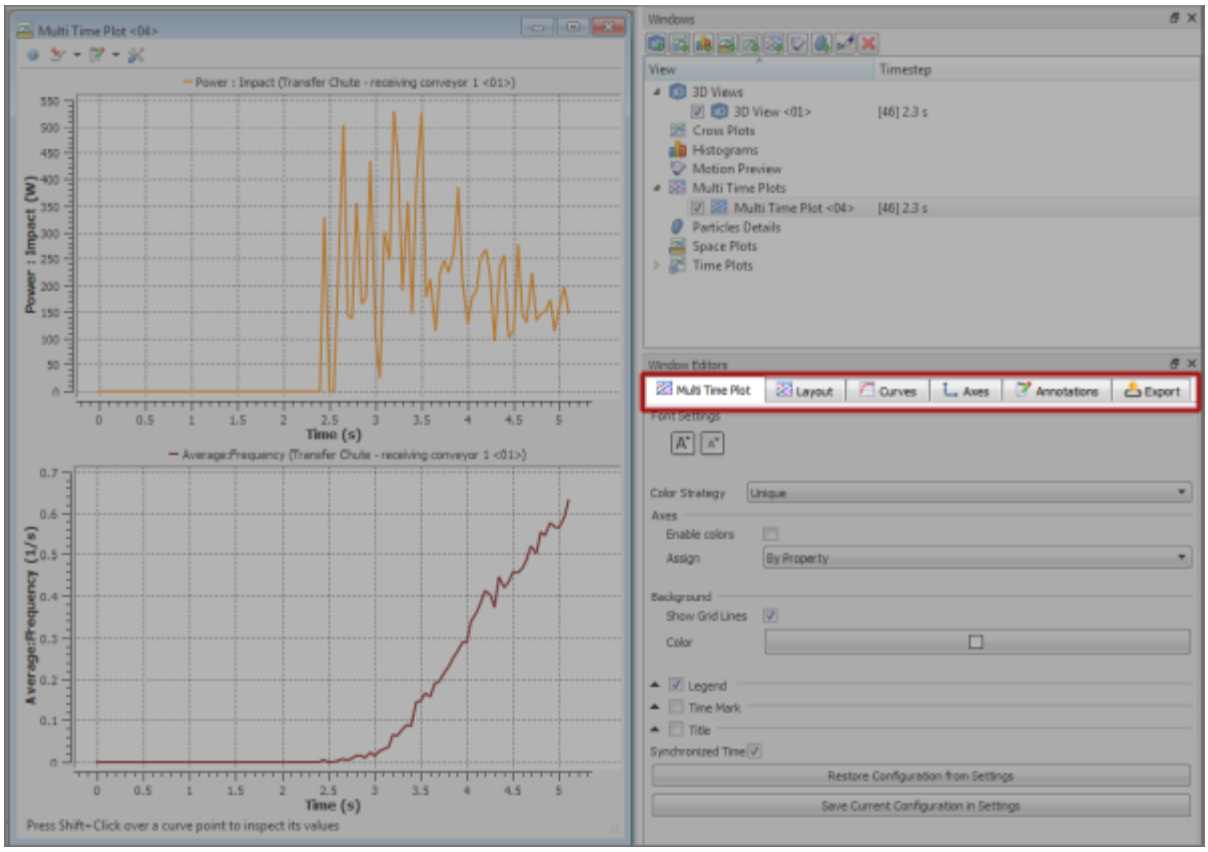
- Changing the Appearance of a Graph (Plot or Histogram) (p. 1125)
- Graphing (Plot or Histogram) a Data Set Within Rocky (p. 1101)

ABOUT CHANGING THE APPEARANCE OF A GRAPH (PLOT OR HISTOGRAM)

After you create your initial plot or histogram (see About Graphing (Plot or Histogram) a Data Set Within Rocky (p. 1101)), you can modify the appearance of your graph by using the Window Editors panel (Figure 5.97: Window Editors panel tabs for a Multi Time Plot (p. 1127)). There are some differences in settings between the various types of graphs, but the manner in which you modify the appearance of these graphical windows is very similar.

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

Figure 5.97: Window Editors panel tabs for a Multi Time Plot



In this topic, we will cover all of the following tabs on the Window Editors panel:

- The "Graph Type" tab
- The Curves tab
- The Axes tab
- The Annotations tab (for Time Plots and Multi Time Plots)
- The Layout tab (for Multi Time Plots)

Note:

For information on the Export tab, see Export Data and Images Out of Rocky (p. 1154).

THE "GRAPH TYPE" TAB

The label on the first tab in the Window Editors panel changes to reflect the type of graph you have selected. On this tab, you may change settings that have to do with the appearance of the graph itself, include the font and background colors, and whether or not the current time step is reflected on your graph or not.

See the image and table below to help you understand how to modify the appearance of a graph by using the first tab on the Window Editors panel.

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

Figure 5.98: Time Plot settings in the Window Editors panel

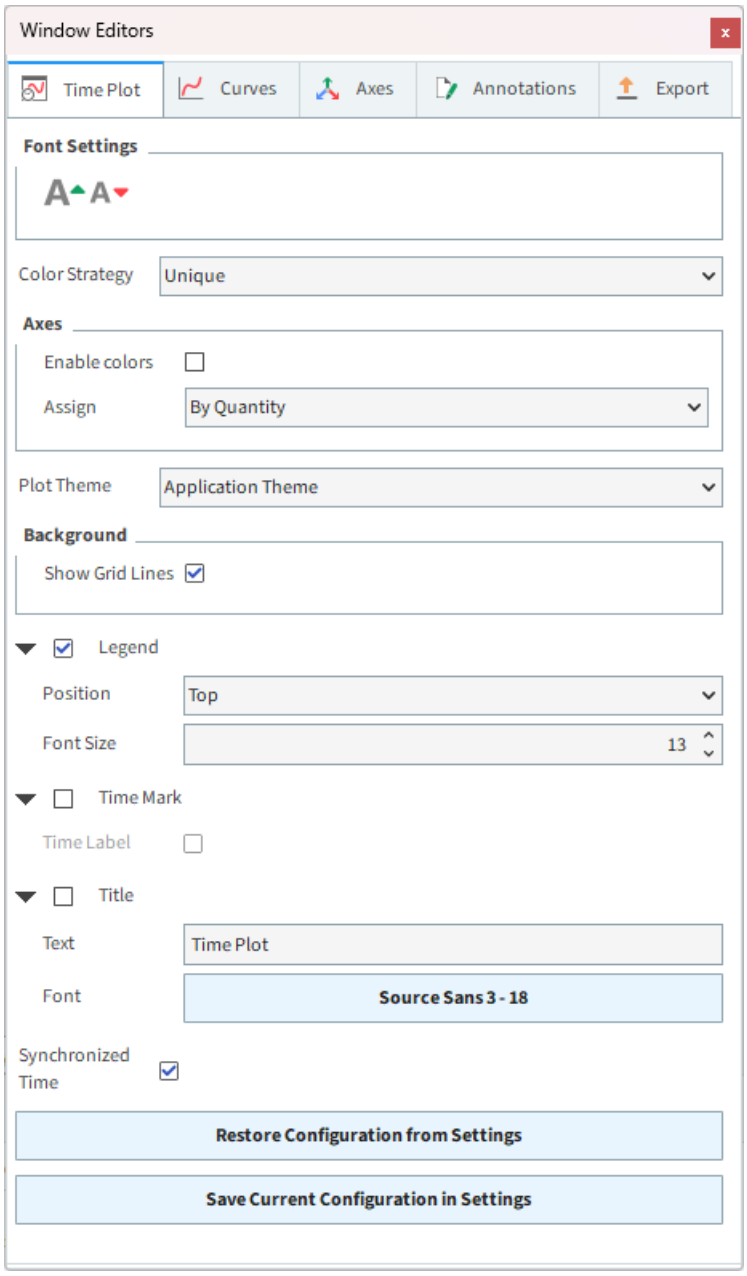


Table 1: Graph (plot and histogram), first tab settings in the Window Editors panel

Setting	Description	Range
Font Settings	The following two buttons in this section apply to all axes, legend, and title text shown in the graph: • **Increase font size**: Makes text one font-size bigger than current display. • **Reduce font size**: Makes text one-font-size smaller than current display. Notes:	Increase font size; Reduce font size

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

Setting	Description	Range
	• These buttons do not affect the text for Annotations. (See the **Font** setting under **Annotations** | **Text** for more information. • You can also change the size of the legend text, the title text, and the axis title and value label text by using those attributes' own **Font** settings. (See further down this table for more information.)	
Color Strategy	Changes the coloring criteria of the curves, which affects the legend text and plot line of the properties (Property or Curves) being graphed. This option can be used to group curves with a similar representation by color based upon the following options: • **Entity Based**: Indicates the geometry, Particle set, or user process upon which the data is based. • **Property Based**: Indicates the Property or Curve upon which the data is based. (See also Use Properties and Curves (p. 931).) • **Unique**: Indicates all available color strategy information including entity, and property based.	Entity Based; Property Based; Unique
Axes Enable colors	Enabling this checkbox colors the Y axes according to the **Color Strategy** selected. Clearing this checkbox keeps all axes colored black.	Turns on or off
Axes Assign	For plots with multiple Properties or Curves, this enables the Y axes to be shared according to the following options: • **By Quantity**: Any properties with a similar type of measurement—no matter the unit assigned—(for example, length, volume, or pressure) will share the same axis. **Note:** Since SPH and CFD Temperatures belong to a different category than DEM Temperatures, these cannot share the same axis. • **By Property**: Each unique property will have its own axis on the plot.	By Quantity; By Property

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

Setting	Description	Range
Plot Theme	Allows you to change the Plots Theme without changing the Application Theme. You chan choose between the current Application Theme, Light Theme, or Dark Theme.	Application Theme; Light Theme; Dark Theme
Background Show Grid Lines	Selecting this checkbox enables black gridlines to appear behind your graph.	Turns on or off
Legend	Enables the legend information, the content and color of which are determined by the **Color Strategy** setting, to be displayed on the graph window.	Turns on or off
Legend Position	When **Legend** is enabled, determines where the Legend information appears on your graph.	Bottom; Left; Right; Top
Legend Font Size	Enables you to change the size of the font used for displaying the Legend text. **Note:** The size you choose will still be affected by the **Font Settings** buttons.	Options limited by the **Select Font** dialog
Time Mark	For Time and Multi Time Plots only, enables a dotted, vertical line to be displayed at the location of the current time step.	Turns on or off
Time Label	When **Time Mark** is selected, enables you to add a label to the mark displaying the current output time value. **Note:** The value displayed reflects the units used for the Time (X) axis.	Turns on or off
Title	Enables you to add a separate title to the top of the graph.	Turns on or off
Title Text	When **Title** is selected, determines what text will be displayed in the title.	No limit
Title Font	Enables you to change the type, style, size, and effects of the font used for displaying the Title text. **Note:** The size you choose will still be affected by the **Font Settings** buttons.	Options limited by the **Select Font** dialog

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

Setting	Description	Range
Synchronized Time	When enabled and the plot window (or any other window with this checkbox enabled) is selected, the details shown in this (and any other Synchronized Time) window will be updated when the current time step is changed on the Time toolbar. (See also About the Time Toolbar (p. 1279).) When cleared and the window is selected, only this window will be updated when the current time step is changed on the Time toolbar. **Tip:** To keep the time step synchronized between multiple plot or other display windows, ensure each window has this option enabled.	Turns on or off
Restore Configuration from Settings	Clicking this button replaces the values set on the tab with the ones that have been saved to the (internal) Rocky Settings folder. (See also Rocky File Types and Folders (p. 129))	(Button selection)
Save Current Configuration in Settings	Clicking this button overwrites the values that have been saved to the (internal) Rocky Settings folder with the ones currently set on the tab. (See also Rocky File Types and Folders (p. 129))	(Button selection)

THE CURVES TAB

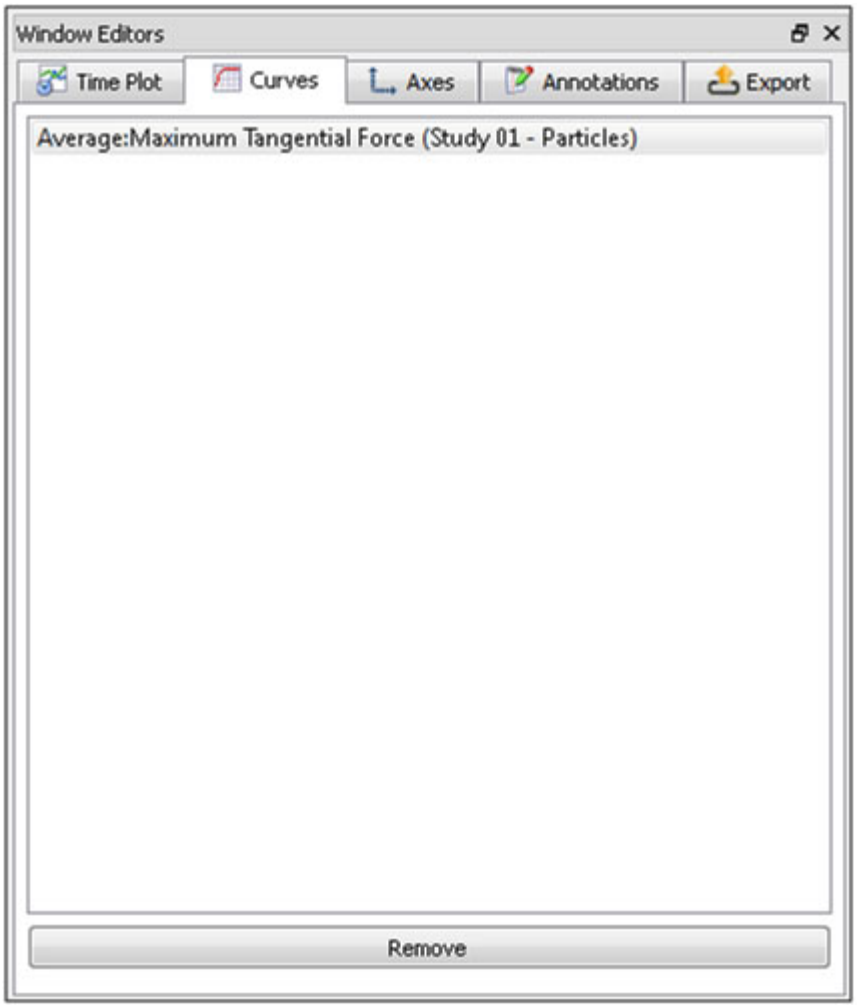
The Curves tab is where you can see what properties, from either the Property or Curves tab, you have added to the selected graph. From here, you can review and/or select and remove these properties to verify or change what data is displayed in your graph (as shown below).

Note:

While you can remove properties from the Curves tab, you must still add them from the Data Editors panel. See also Graph Data within Rocky by Creating a Plot or Histogram (p. 1124).

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

Figure 5.99: Time Plot, Curves display in the Window Editors panel



THE AXES TAB

The Axes tab enables you to change display settings related only to the various axes that make up your graph, including axis titles, value ranges, and scales.

See the image and table below to help you understand how to modify the display of your axes by using the Axes tab on the Window Editors panel.

Tip:

You can also access these options by double-clicking an axis on your plot window.

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

Figure 5.100: Time Plot, Axes settings in the Window Editors panel

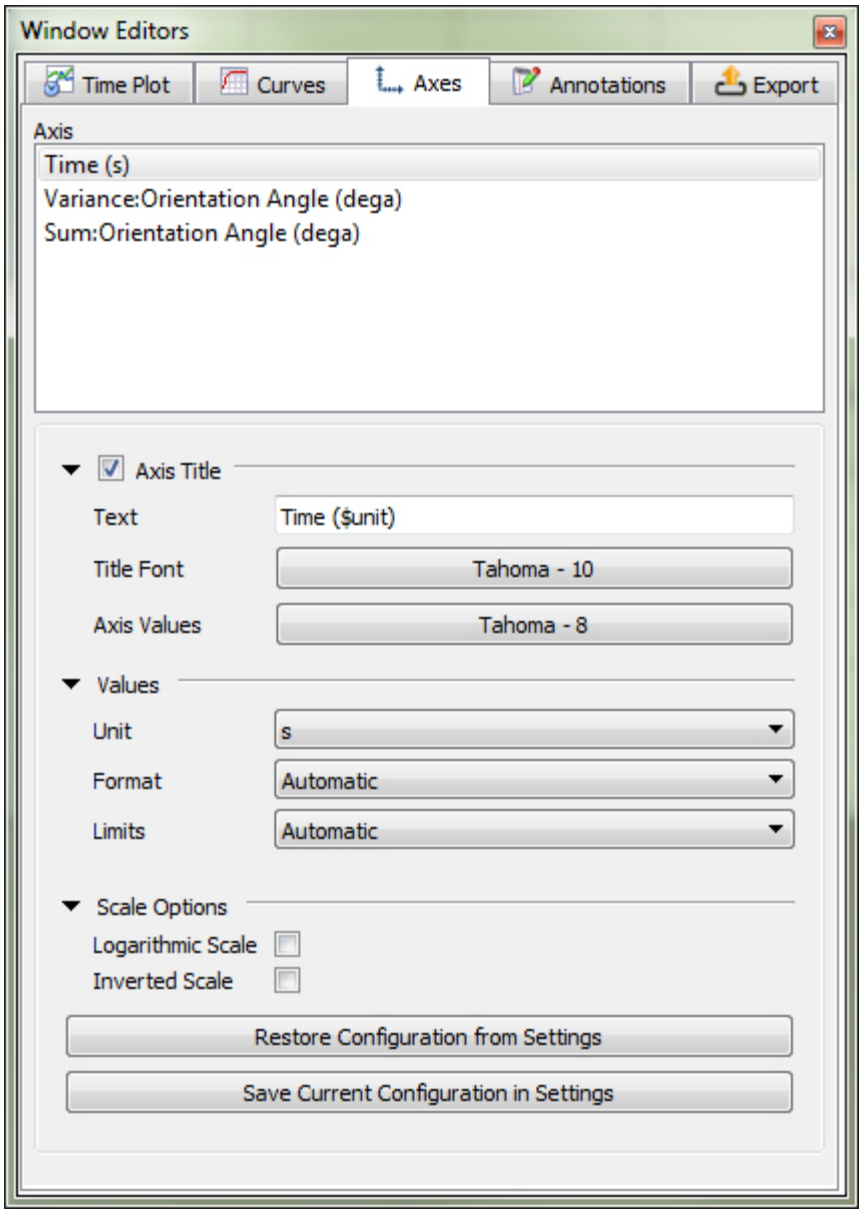


Table 2: Graph (plot and histogram), Axes tab settings in the Window Editors panel

Setting	Description	Range
Axes	Lists each separate axis that has been added to the graph.	List generated automatically
Axis Title	When enabled, adds a title labeling the selected axis.	Turns on or off
Axis Title Text	When **Axis Title** is selected, enables you to modify the default text labeling the selected axis.	No limit

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

Setting	Description	Range
Axis Title Font	Enables you to change the type, style, size, and effects of the font used for displaying the title **Text** for the selected axis. **Note:** The size you choose will still be affected by the **Font Settings** buttons located on the first tab of the Window Editors panel.	Options limited by the **Select Font** dialog
Axis Title Axis Values	Enables you to change the type, style, size, and effects of the font used for displaying the value marks for the selected axis. **Note:** The size you choose will still be affected by the **Font Settings** buttons located on the first tab of the Window Editors panel.	Options limited by the **Select Font** dialog
Values Unit	Enables you to change the units for which the values for the selected axis are displayed.	Automatically provided
Values Format	Enables you to change the number of decimal places in which you want the axis values displayed. Specifically: • **Automatic**: Enables Rocky to choose the optimal decimal places to display based upon the data shown. • **Integer**: Rounds values to the nearest whole integer. (No decimal places.) • **Decimal X digits**: Displays X number of digits after the decimal. • **Exponential X digits**: Displays X number of digits after the decimal, power required to express the value as an exponential number.	Integer; Decimal 2 digits; Decimal 4 digits; Decimal 8 digits; Exponential 2 digits; Exponential 4 digits; Exponential 8 digits; Automatic
Values Limits	Sets how the minimum and maximum values for the selected axis are determined. Specifically: • **Automatic**: Enables Rocky to optimize the limits shown. based upon the data • **User Defined**: Enables you to enter custom limit values.	Automatic; User Defined

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

Setting	Description	Range
Values Min	When **Limits** is set to **User Defined**, this enables you to set a custom minimum value for the selected axis.	No limit
Values Max	When **Limits** is set to **User Defined**, this enables you to set a custom maximum value for the selected axis.	No limit
Values Step	When **Limits** is set to **User Defined**, this enables you to set the amount you want between each value mark on the selected axis.	No limit
Scale Options Logarithmic Scale	When enabled, this converts the scale of the selected axis to logarithmic. When cleared, the scale is a standard linear one.	Turns on or off
Scale Options Inverted Scale	When enabled, this swaps the minimum and maximum value placements on the selected axis.	Turns on or off
Restore Configuration from Settings	Clicking this button replaces the values set on the tab with the ones that have been saved to the (internal) Rocky Settings folder. (See also Rocky File Types and Folders (p. 129))	(Button selection)
Save Current Configuration in Settings	Clicking this button overwrites the values that have been saved to the (internal) Rocky Settings folder with the ones currently set on the tab. (See also Rocky File Types and Folders (p. 129))	(Button selection)

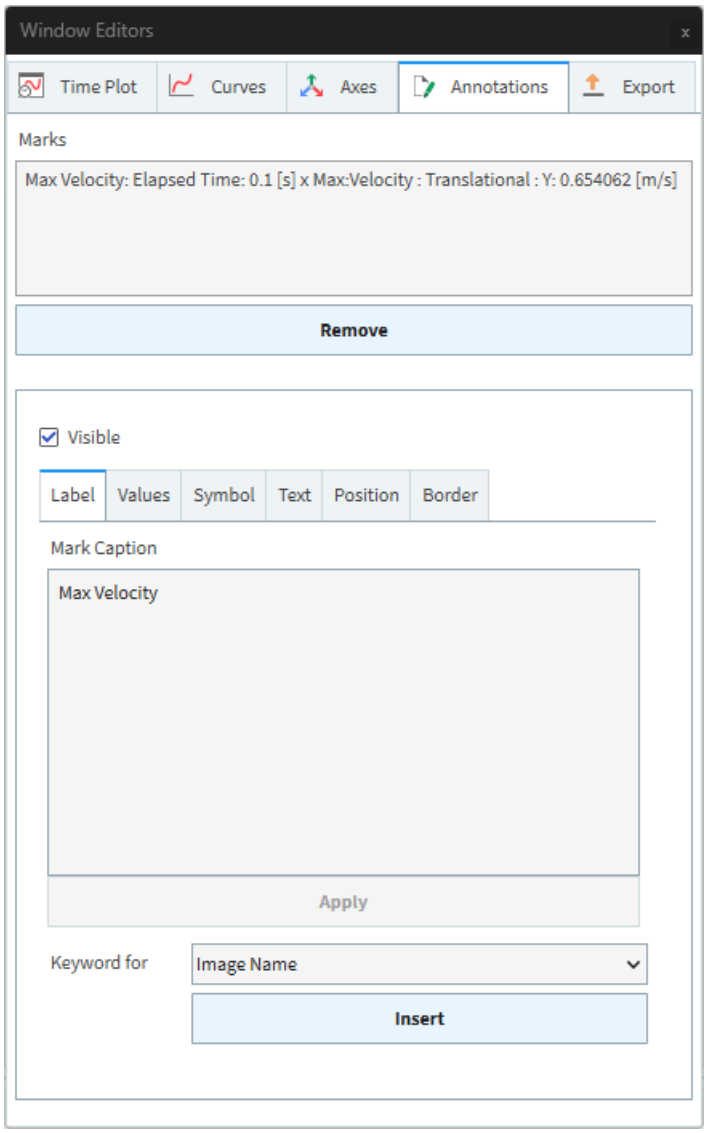
THE ANNOTATIONS TAB

The Annotations tab is where you edit the values and change the display settings for the Annotation marks that you have already added to either a Time Plot or a Multi Time Plot. (See also About Creating an Annotation on a Time Plot or Multi Time Plot (p. 1145).)

See the image and table below to help you understand how to edit your Annotations by using the Annotations tab on the Window Editors panel.

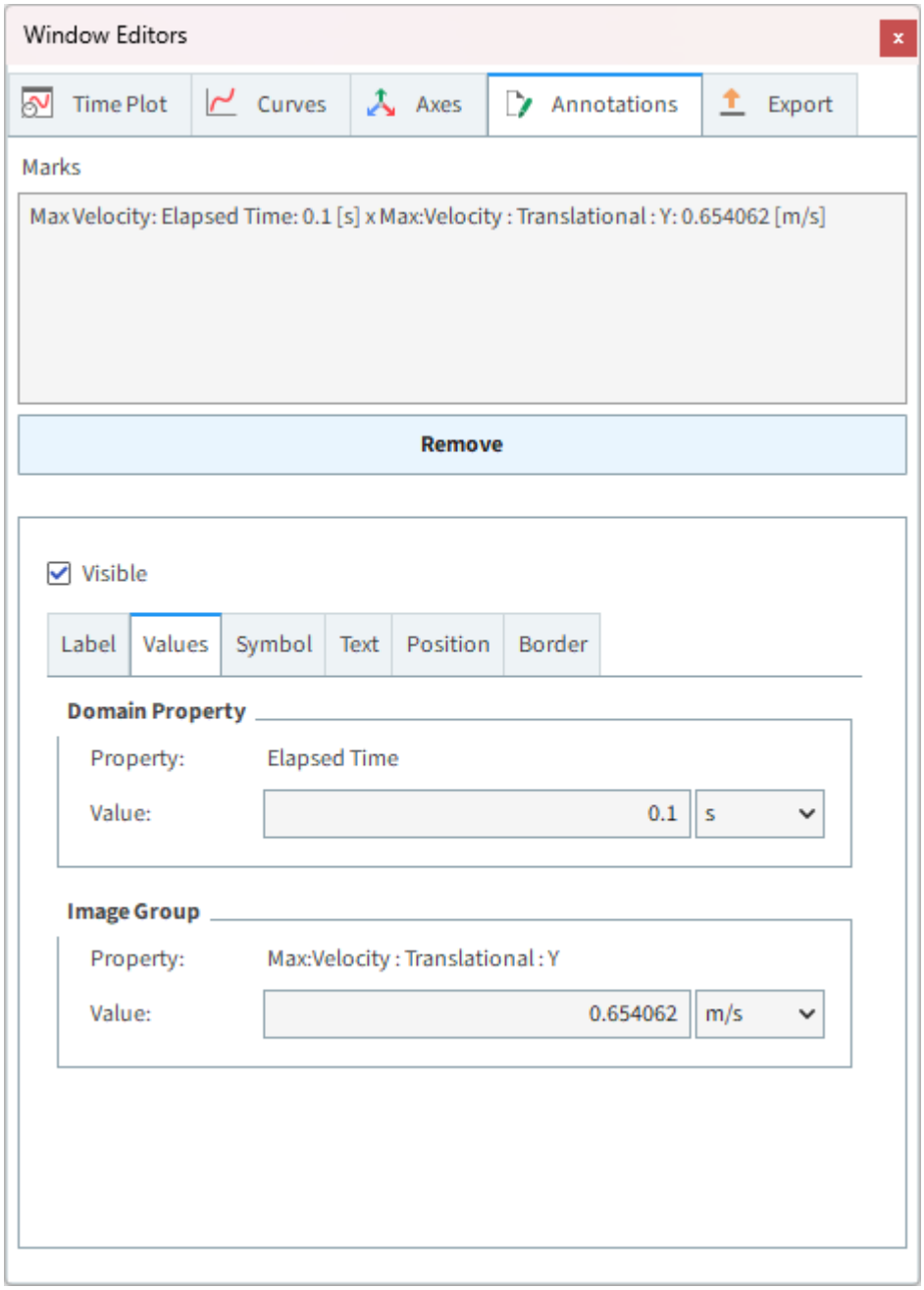
Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

Figure 5.101: Time Plot, Annotations | Label settings in the Window Editors panel



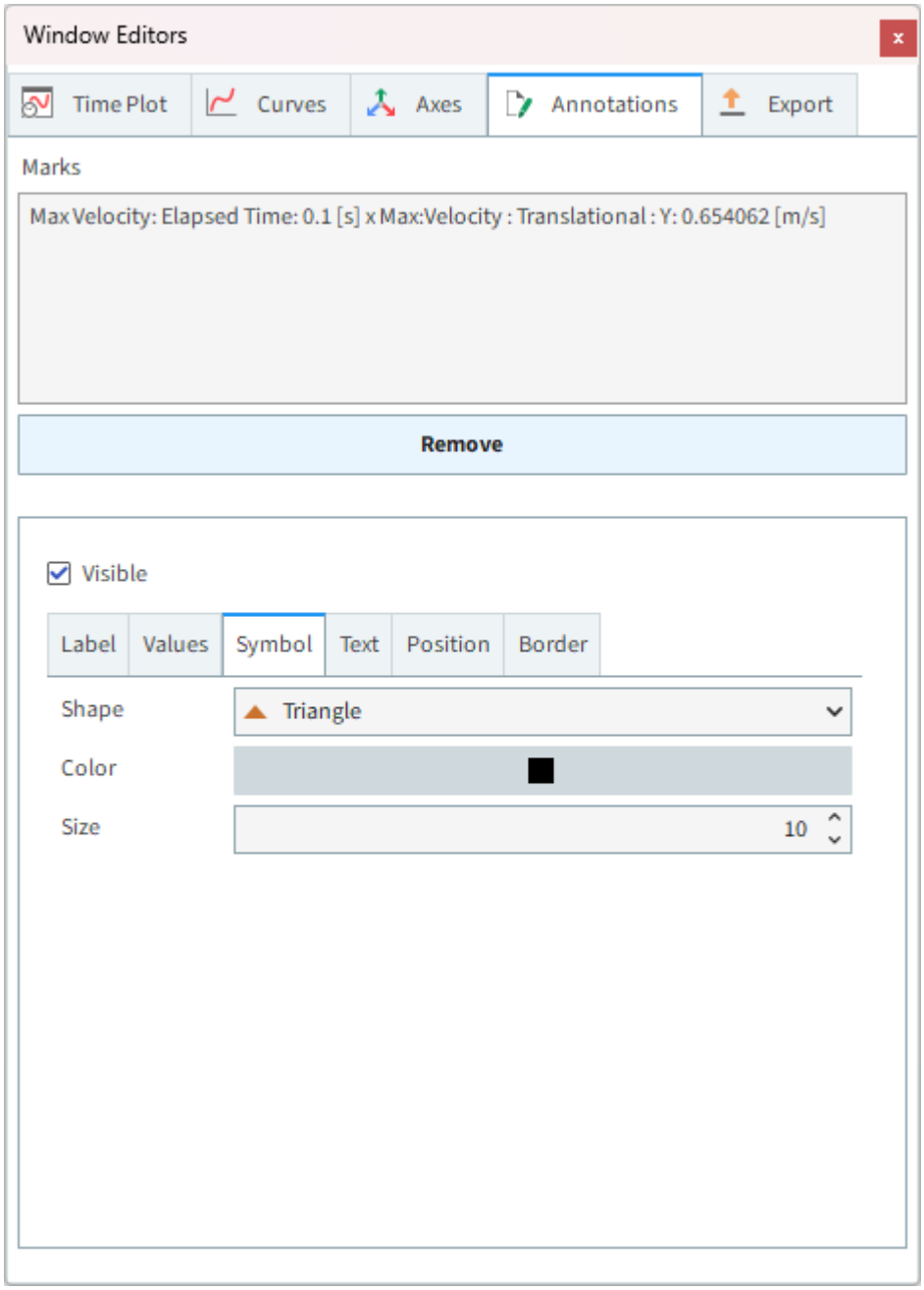
Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

Figure 5.102: Time Plot, Annotations | Values settings in the Window Editors panel



Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

Figure 5.103: Time Plot, Annotations | Symbol settings in the Window Editors panel



Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

Figure 5.104: Time Plot, Annotations | Text settings in the Window Editors panel

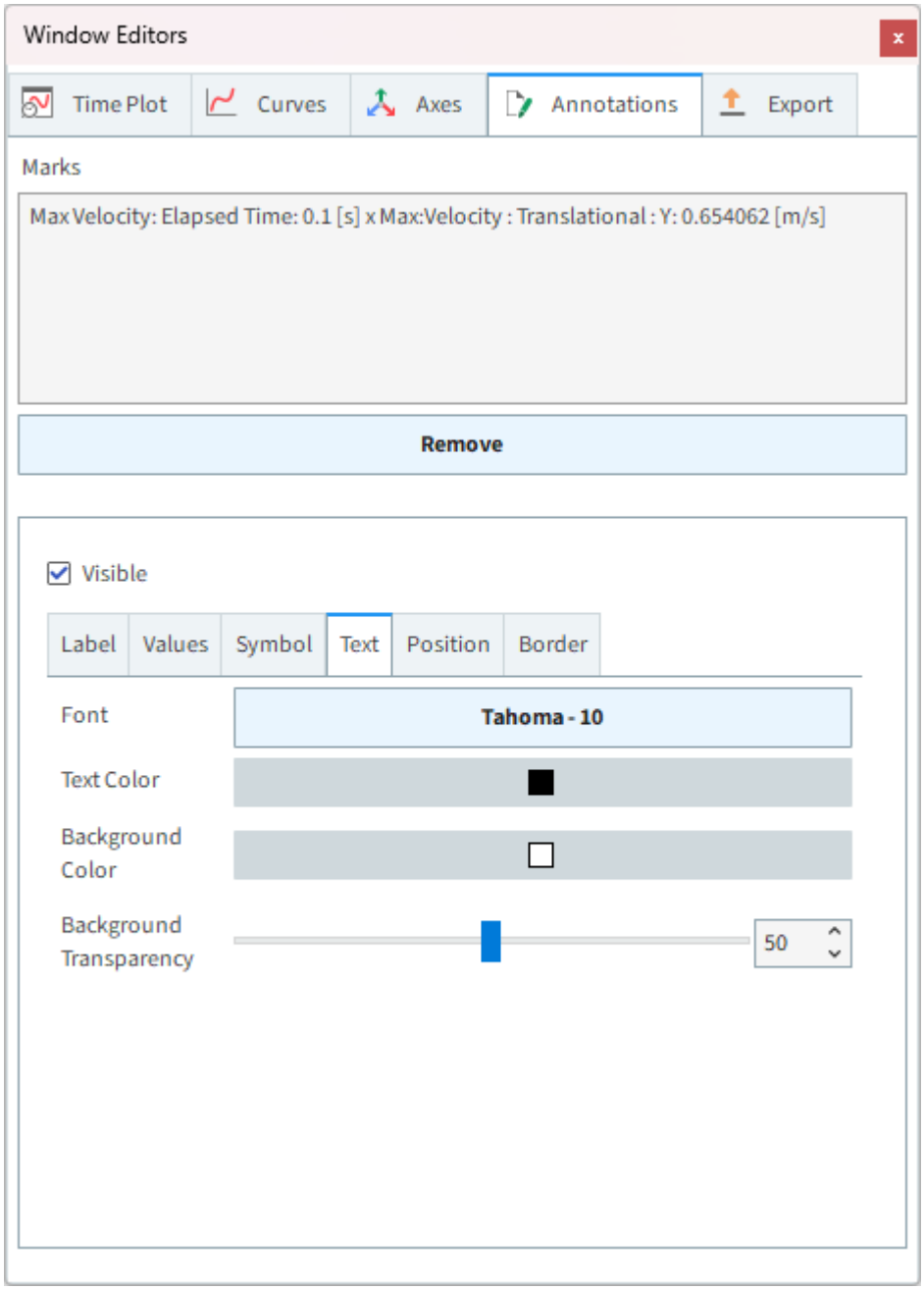
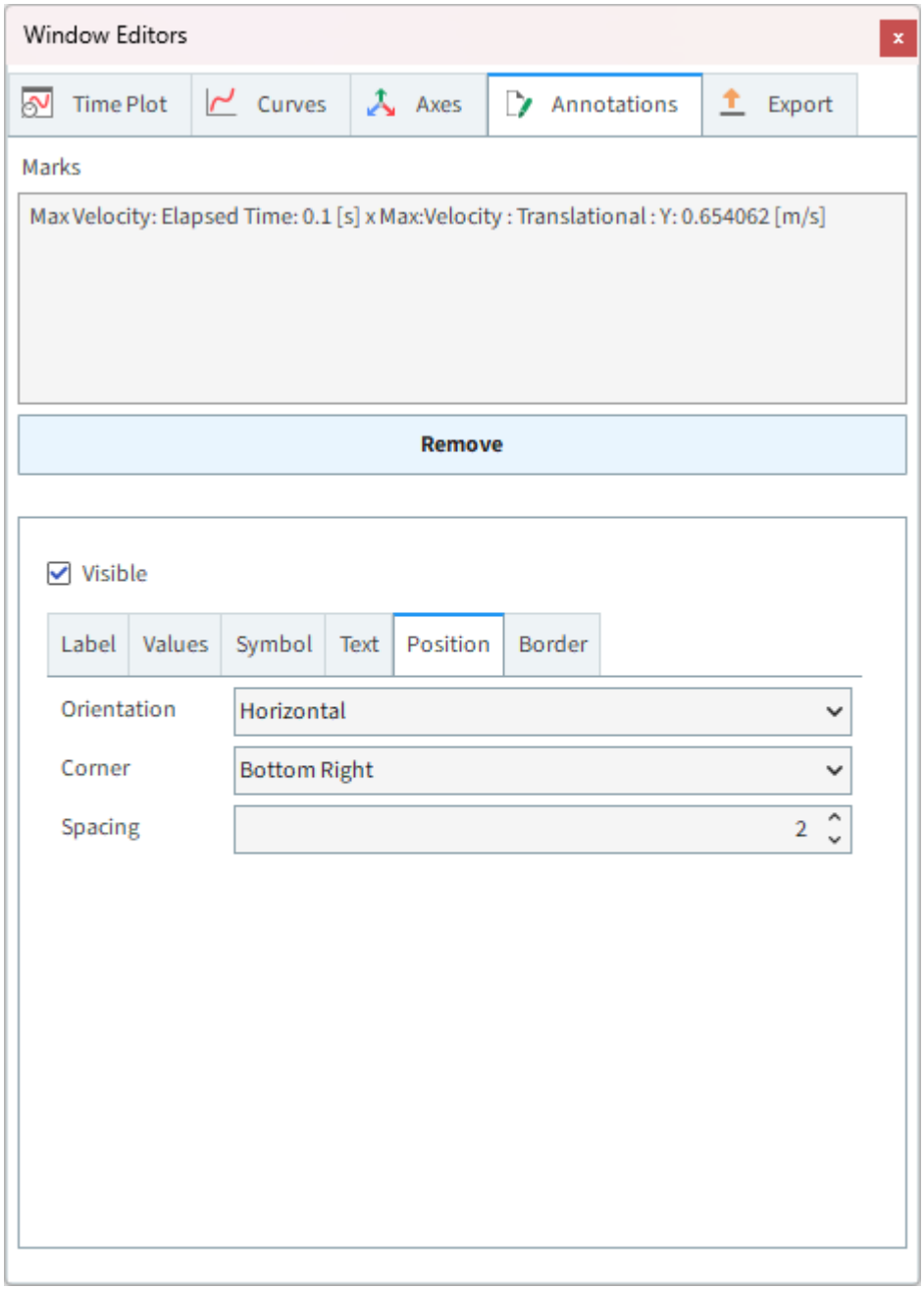


Figure 5.105: Time Plot, Annotations | Position settings in the Window Editors panel



Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

Figure 5.106: Time Plot, Annotations | Border settings in the Window Editors panel

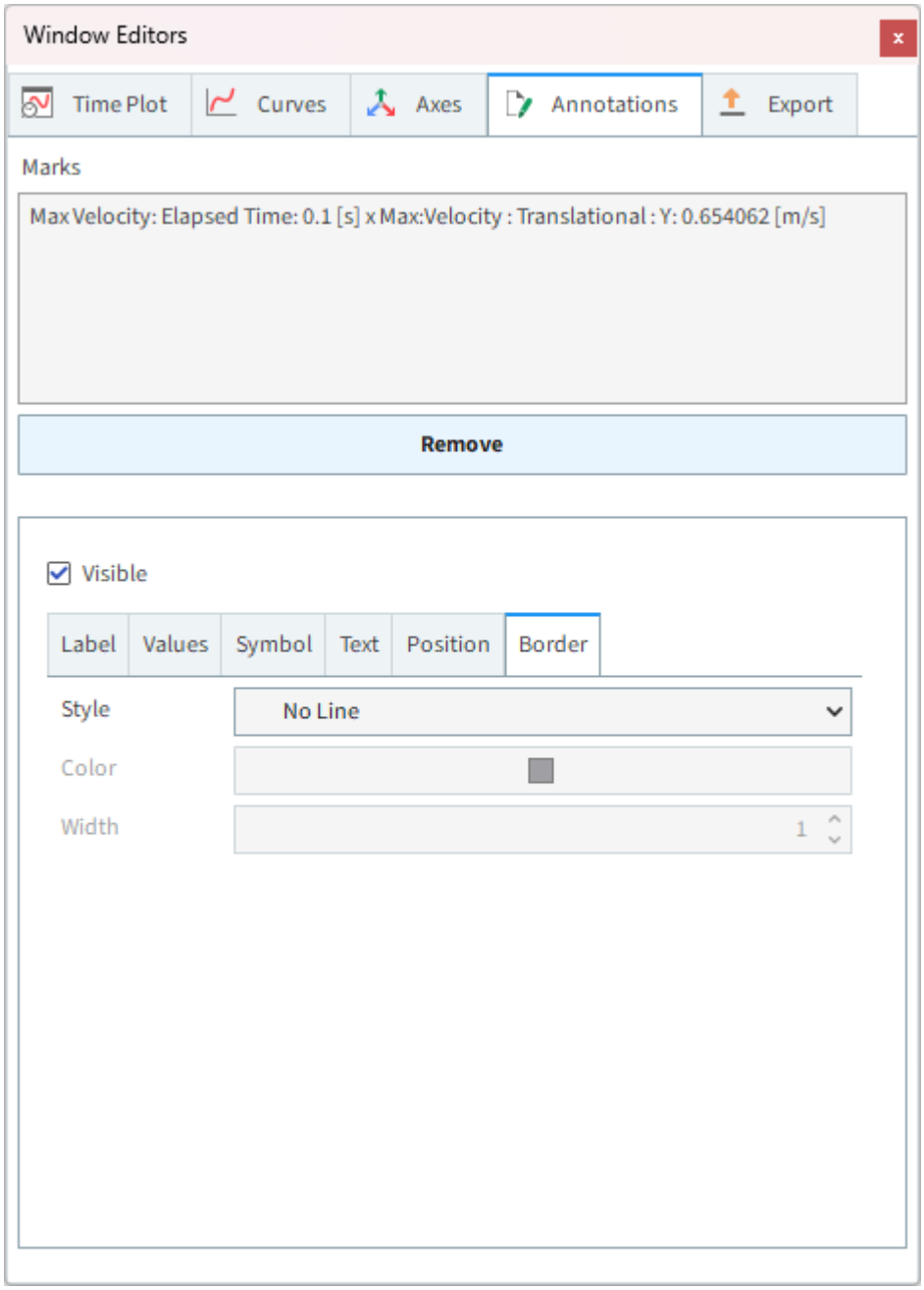


Table 3: Time Plot and Multi Time Plot, Annotations tab settings in the Window Editors panel

Setting	Description	Range
Marks	Lists each separate Annotation that has been added to the graph. This is where you select which Annotation you want to modify.	List generated automatically
Remove	Allows you to remove an Annotation.	
Visible	When enabled, this shows the selected Annotation on the graph. When cleared, this hides the selected Annotation.	Turns on or off

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

Setting	Description	Range
Label		
Mark Caption	Enables you to modify what text is displayed on the graph for the selected Annotation. **Tip:** Use the Enter key to indicate where you want line breaks.	No limit
Keyword for	Enables you to select a dynamic keyword from the dropdown list. The keywords display information from your simulation.	Image Name; Image Unit; Image Value; Image Max Value; Image Min Value; Domain Value; Domain Unit; Study Name; Study Description; Customer Name; Application Version; Input Variables
Insert	Inserts the keyword you selected into the Mark Caption field.	No limit
Apply	Applies the changes made in the Mark Caption field.	No limit
Values		
Domain Property | Property	Displays the property you chose in the Create Annotation dialog for the Domain (X axis) value for the selected Annotation. **Note:** This setting is not editable.	Display only
Domain Property | Value	Enables you to edit the value you entered for the **Property** domain. This will be X axis location of the selected Annotation mark.	No limit

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

Setting	Description	Range
Image Group | Property	Displays the property you chose in the Create Annotation dialog for the Image (Y axis) value for the selected Annotation. **Note:** This setting is not editable.	Display only
Image Group | Value	Enables you to edit the value you entered for the **Property** domain. This will be Y axis location of the selected Annotation mark.	No limit
Symbol		
Shape	Enables you to change the shape of the symbol used to mark the selected Annotation. Besides the five shape symbols listed, you can also select **No Symbol** to have only the Annotation caption text be displayed on the graph.	Ellipse; Rectangle; Diamond; Triangle; Down Triangle; No Symbol
Color	Enables you to change the color of the symbol used to mark the selected Annotation.	Options limited by the **Select Color** dialog
Size	Enables you to change the size of the symbol used to mark the selected Annotation.	Whole, positive values
Text		
Font	Enables you to change the type, style, size, and effects of the font used for displaying the **Mark Caption** text for the selected Annotation. **Note:** The size you choose will not be affected by the **Font Settings** buttons located on the first tab of the Window Editors panel.	Options limited by the **Select Font** dialog
Text Color	Enables you to change the color of the font used for displaying the **Mark Caption** text for the selected Annotation.	Options limited by the **Select color** dialog
Background Color	Enables you to change the color appearing behind the text for the selected Annotation.	Options limited by the **Select color** dialog
Background Transparency	Enables you to change the transparency level of the **Background Color** selected. The higher the value, the more transparent the color.	0-100
Position		
Orientation	Enables you to change the orientation of the text for the selected Annotation.	Horizontal; Vertical

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

Setting	Description	Range
Corner	Enables you to set where in relation to the mark to place the text for the selected Annotation.	Top Left; Top; Top Right; Left; Center; Right; Bottom Left; Bottom; Bottom Right
Spacing	Enables you to set how many pixels away from the mark to place the text for the selected Annotation.	Whole numbers greater than or equal to one
Border		
Style	Enables you to set what kind of border you want surrounding the text for the selected Annotation. If you want no border, choose **No Line**.	No Line; Solid Line; Dash Line; Dot Line; Dash Dot Line; Dash Dot Dot Line
Color	When you select a **Style** other than **No Line**, this enables you to change the color of the border.	Options limited by the **Select Color** dialog
Width	When you select a **Style** other than **No Line**, this enables you to change the width of the border.	Whole numbers greater than or equal to one

THE LAYOUT TAB

This tab appears only for Multi Time Plots and enables you to change how the various sub-plots are displayed, including the number of columns and the minimum sub-plot dimensions within the graph window.

See the image and table below to help you understand how to modify layout of your Multi Time Plots by using the layout tab on the Window Editors panel.

Figure 5.107: Multi Time Plot, Layout settings in the Window Editors panel

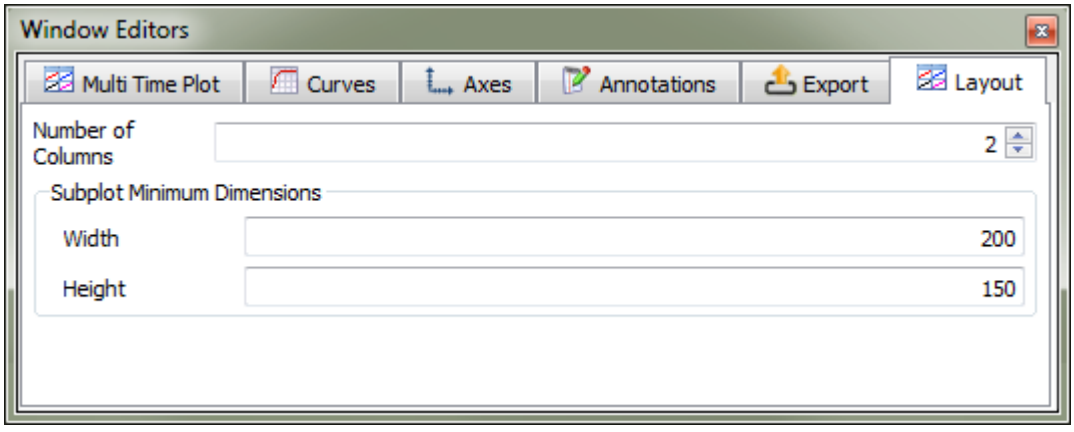


Table 4: Multi Time Plot, Layout tab settings in the Window Editors panel

Setting	Description	Range
Number of Columns	Enables you to set how many columns into which you want the sub-plots of your selected Multi Time Plot arranged.	Whole numbers greater than or equal to one

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

Setting	Description	Range
Subplot Minimum Dimensions Width	Enables you to define the minimum pixel width you want Rocky to maintain for each individual sub-plot in the selected Multi Time Plot. When making the window size smaller, the sub-plots will continue to be displayed in full until this setting is reached, at which point scroll bars will be added.	Whole values greater than or equal to 300
Subplot Minimum Dimensions Height	Enables you to define the minimum pixel height you want Rocky to maintain for each individual sub-plot in the selected Multi Time Plot. When making the window size smaller, the sub-plots will continue to be displayed in full until this setting is reached, at which point scroll bars will be added.	Whole values greater than or equal to 150

What would you like to do?

- Use the Window Editors Panel to Change the Appearance of a Graph (Plot or Histogram) (p. 1149)
- Graph Data within Rocky by Creating a Plot or Histogram (p. 1124)

See Also:

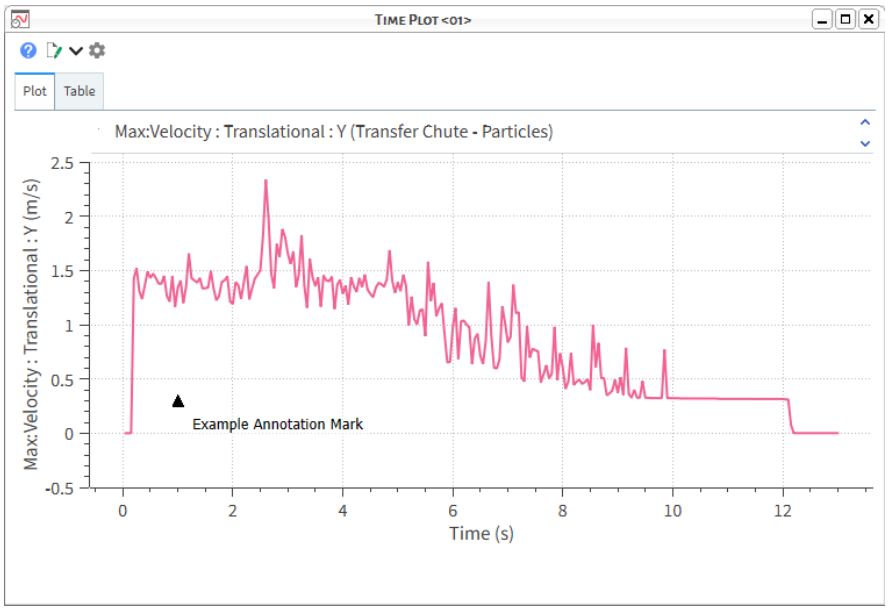
- Graph (Plot or Histogram) a Data Set Within Rocky (p. 1101)
- Double-Click a Plot Line, Bar, or Point to Change Its Appearance (p. 121)

ABOUT CREATING AN ANNOTATION ON A TIME PLOT OR MULTI TIME PLOT

An Annotation is a special mark that you can add to a Time Plot or a Multi Time Plot that shows a particular data point and caption that you define (as shown below). This is useful in cases where you want to highlight a particular event on the graph to help provide context. For example, you might add an Annotation at the moment when the particle flow reached steady state, when a hopper valve was opened, or when you have reached the maximum mass of particles inside a particular region.

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

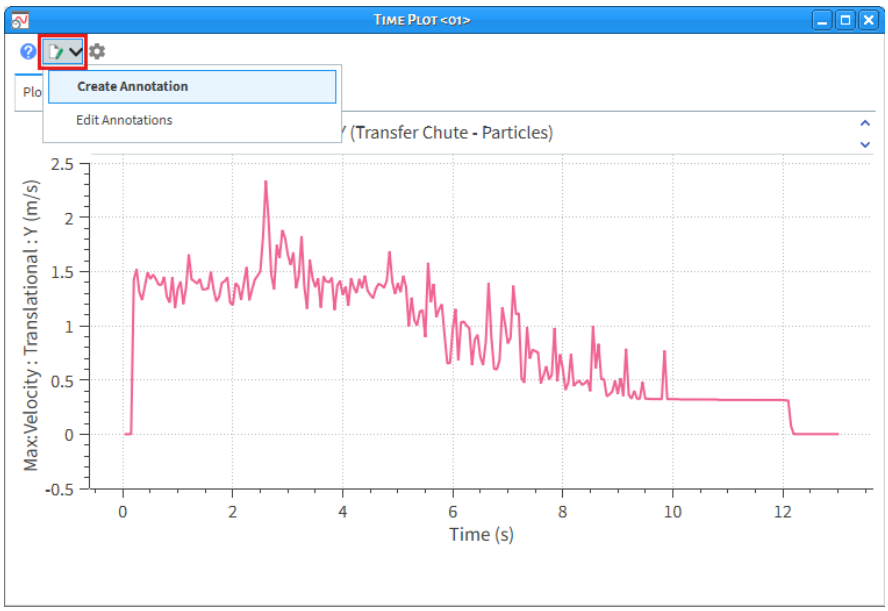
Figure 5.108: Example Annotation added to a Time Plot



You add the initial Annotation and define the text and values from a separate menu button on the graph itself (as shown below) but then can change the display options for it, including settings like font size and position, mark shape, and so on, on the Annotations tab of the Window Editors panel. (See also About Changing the Appearance of a Graph (Plot or Histogram) (p. 1126)).

See the images and table below to understand how to create an Annotation for a Time Plot or Multi Time Plot.

Figure 5.109: Location of the Annotations menu on the graph window



Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

Figure 5.110: Create Annotation dialog

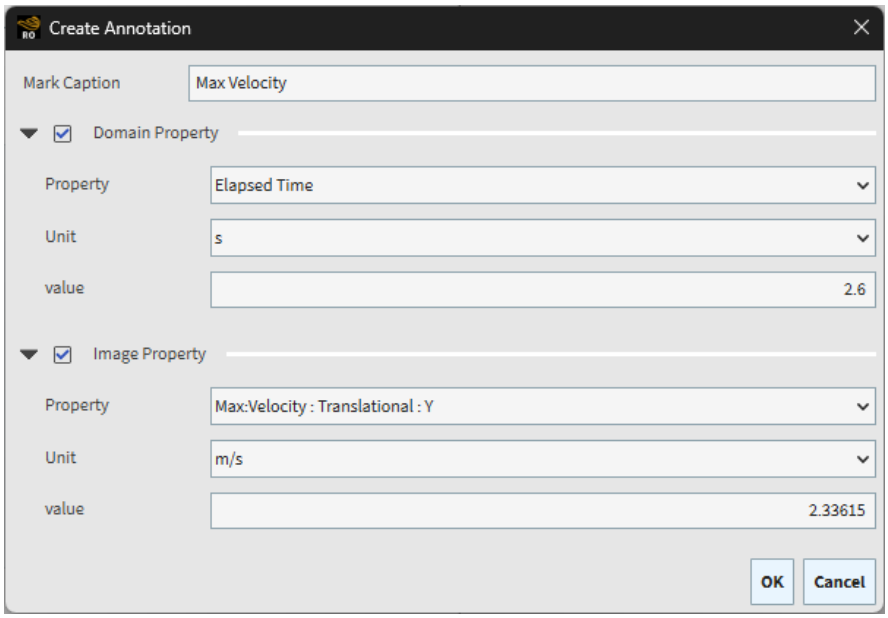


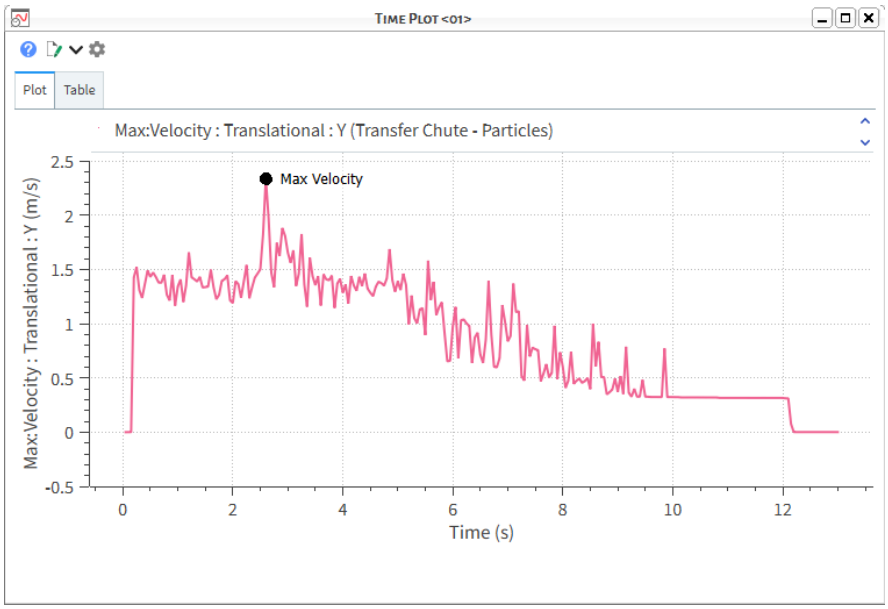
Table 1: Create Annotation dialog options

Setting	Description	Range
Mark Caption	Enables you to enter what text is displayed on the graph for the new Annotation. **Note:** You can add line breaks in the Annotations tab on the Window Editors panel.	No limit
Domain Property	Settings defined by the Property or Curve graphed for the domain (X) axis. For Time Plots and Multi Time Plots, the domain is always Time. **Note:** This setting does not affect how the Annotation is created. Even if cleared, Domain Property values will still be used in the Annotation definition.	Turns on or off
Domain Property Property	The Property or Curve graphed for the domain (X) axis to which you want the Annotation applied. For Time Plots and Multi Time Plots, the only choice is Elapsed Time.	Elapsed Time
Domain Property Unit	Enables you to set the units for which the **value** of the **Property** is defined.	Automatically provided

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

Setting	Description	Range
Domain Property value	Enables you to set where along the domain (X) axis to place the Annotation marker.	No limit
Image Property	Settings defined by the Properties or Curves graphed for the image (Y) axis. **Note:** This setting does not affect how the Annotation is created. Even if cleared, Image Property values will still be used in the Annotation definition.	Turns on or off
Image Property Property	The Property or Curve graphed for the image (Y) axis to which you want the Annotation applied.	Automatically provided
Image Property Unit	Enables you to set the units for which the **value** of the **Property** is defined.	Automatically provided
Image Property value	Enables you to set where along the image (Y) axis to place the Annotation marker.	No limit

Figure 5.111: Annotation Created



What would you like to do?

- Create an Annotation on a Time Plot or Multi Time Plot (p. 1149)
- Change the Appearance of an Annotation for a Time Plot or Multi Time Plot (p. 1126)

See Also:

- Graph (Plot or Histogram) a Data Set Within Rocky (p. 1101)

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

USE THE WINDOW EDITORS PANEL TO CHANGE THE APPEARANCE OF A GRAPH (PLOT OR HISTOGRAM)

1. Ensure the **Window Editors** panel is visible. (From the **View** menu, click **Window Editors**.)
2. From the Workspace (p. 1396), select the plot or histogram window that you want to change. (See also Graph Data within Rocky by Creating a Plot or Histogram (p. 1124).)
3. From the **Window Editors** panel, select the options you want on the "Graph Type", Curves, Axes, and Annotations tabs. (See also About Changing the Appearance of a Graph (Plot or Histogram) (p. 1126).) Any changes you make are automatically updated in the selected window.

Tips:

- You can also open the **Window Editors** panel for a window by right-clicking an empty space within the plot or histogram window (for example, the background behind a curve), and then clicking **Settings**.
- To change how data is weighted in a Histogram, use the **Configure Histogram** dialog. (See also Configure how Data is Displayed in a Histogram (p. 1151)).
- To change the theme of your Plot, you can use the **Plot Theme** option in the **Window Editors** panel and choose the one you prefer.
- If changes to your graph take too long to update, you can either minimize the window or create a new workspace for the window. This will save changes to the graph without causing the updates to affect your other work in Rocky.
- To reuse in future projects the settings you made on either the "Graph Type" or Axes tabs, click the **Save Current Configuration in Settings** button on the tab containing the values you want to retain.
- To apply settings you have already saved (either by previously using the **Save Current Configuration in Settings** button or by saving selections within the **Preferences** dialog) to either the "Graph Type" or Axes tabs, click the **Restore Configuration from Settings** button on the tab containing the values you want to replace.

See Also:

- About Changing the Appearance of a Graph (Plot or Histogram) (p. 1126)
- About Histograms (p. 1112)
- Configure how Data is Displayed in a Histogram (p. 1151)
- Graph (Plot or Histogram) a Data Set Within Rocky (p. 1101)

CREATE AN ANNOTATION ON A TIME PLOT OR MULTI TIME PLOT

1. From the Workspace (p. 1396), select the Time Plot or Multi Time Plot window to which you want to add an Annotation. (See also Graph Data within Rocky by Creating a Plot or Histogram (p. 1124).) Important Ensure you have already added the properties (Properties and/or Curves) that you want to the graph.

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

2. From the top left corner of the window, click the arrow next to the **Select the types of annotations to display in the plot.** button, and then click **Create Annotation**.
3. From the **Create Annotation** dialog, do all of the following:
   a. In the **Mark Caption** field, enter the Annotation text you want to display on the graph.
   b. Under **Domain Property**, ensure the **Unit** is as you want and then enter the domain axis location of the mark in the **value** field.
   c. Under **Image Property**, select what you want from the **Property** list, verify the **Unit** setting, and then enter the image axis location of the mark in the **value** field.
4. Click **OK**. The Annotation text and mark you defined appears in the graph.

Tip:

To change the appearance, values, and other settings of this newly added Annotation, use the Window Editors panel. (See also About Changing the Appearance of a Graph (Plot or Histogram) (p. 1126).)

Tip:

You can use the **Remove** option in the **Window Editors** panel, under Annotations tab, to remove an Annotation from a Plot.

See Also:

- About Creating an Annotation on a Time Plot or Multi Time Plot (p. 1145)
- Add Dynamic Annotations to a Timeplot or Multi Time Plot (p. 1150)
- Graph (Plot or Histogram) a Data Set Within Rocky (p. 1101)

ADD DYNAMIC ANNOTATIONS TO A TIME PLOT OR MULTI TIME PLOT

1. In the Workspace, select the Time Plot or Multi Time Plot window you want to edit the annotation for.
2. From the top left corner of the window, click the **Settings** button. The **Window Editors** menu becomes available.
3. Select the **Annotations** tab and from the **Marks** option, select the Annotation you wish to edit.
4. From the **Mark Caption** field, do the following:
   a. Add any text you want to have displayed in the caption.
   b. From the **Keyword for** dropdown list, select the keyword that matches the simulation entity you want to display.
   c. After selecting the keyword, click **Insert**.

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

d. You can repeat steps 2 and 3 to insert all the keywords you find necessary.

5. Click **Apply** to finish edditing the text.

Tip:

To change the appearance, values, and other settings of this newly added Annotation, use the Window Editors panel. (See also About Changing the Appearance of a Graph (Plot or Histogram) (p. 1126).)

Tip:

You can use the **Remove** option in the **Window Editors** panel, under Annotations tab, to remove an Annotation from a Plot.

See Also:

- About Creating an Annotation on a Time Plot or Multi Time Plot (p. 1145)
- Create an Annotation on a Time Plot or Multi Time Plot (p. 1149)
- Graph (Plot or Histogram) a Data Set Within Rocky (p. 1101)

CONFIGURE HOW DATA IS DISPLAYED IN A HISTOGRAM

1. From the Workspace (p. 1396), select the histogram window that you want to configure. (See also Graph Data within Rocky by Creating a Plot or Histogram (p. 1124).)
2. In the upper-left corner of the window, click the **Configure histogram** button.
3. In the **Configure Histogram** dialog, do any or all of the following:
   - Select the options you want for **Weight**. Important If you have chosen to display a Property in your Histogram, ensure you choose a weight that is applicable to the Property you chose. (See also About Histograms (p. 1112).)
   - Select the options you want for **Number of Bins**, **Cumulative Bins**, and **Percent Values**.
4. To change the value limits, select the Property or Curve from the **Properties** list, select **User Defined** from the **Limits** list, and then change the **Min** and/or **Max** values as you want.
5. Click **OK** to close the dialog. Note: The Histogram window is automatically updated with your changes as you make them.

Tips:

- To change how your graph appears, use the **Window Editors** panel. (See also Use the Window Editors Panel to Change the Appearance of a Graph (Plot or Histogram) (p. 1149)).
- If changes to your graph take too long to update, you can either minimize the window or create a new workspace for the window. This will save changes to the graph without causing the updates to affect your other work in Rocky.

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

See Also:

- About Histograms (p. 1112)
- About Changing the Appearance of a Graph (Plot or Histogram) (p. 1126)
- Graph (Plot or Histogram) a Data Set Within Rocky (p. 1101)

ADD AND EDIT CUSTOM TABLE FORMULAS IN A TIME PLOT

1. From the Workspace (p. 1396), select the Time Plot window to which you want to add or edit a custom Formula (see also Graph Data within Rocky by Creating a Plot or Histogram (p. 1124), and then select the **Table** tab.
2. Do one of the following:
   - To add a custom curve to the Time Plot, click the **Add Formula** button, enter what you want on the **Add Expression** dialog, and then click **OK**. A new column with the curve data you defined appears in the table and a new curve appears in the plot.
   - To edit a custom curve that you have already added, select the column header of the custom curve you want to change, click the **Edit Formula** button, enter what you want on the **Edit Expression** dialog, and then click **OK**. The curve data is updated both in the table column and on the plot itself.
   - To remove a custom curve that you have already added, select the column header of the custom curve you want to remove, and then click the **Remove Formula** button. The curve data is removed from both the table and the plot.
3. Select what you want for the **Interpolate Values** checkbox.

See Also:

- About Time Plots (p. 1102)
- Graph (Plot or Histogram) a Data Set Within Rocky (p. 1101)

USING THE COLORING TAB TO CHANGE A GRAPH (PLOT OR HISTOGRAM) WINDOW

The Coloring tab, located on the Data Editors panel for all simulation entities and User Processes, enables you to change the colors and data attributes of the entities being displayed in a plot or histogram window.

What would you like to do?

- Learn more About Using the Coloring Tab to Change a Plot or Histogram (p. 1152)
- Use the Coloring Tab to Change a Plot or Histogram (p. 1154)

See Also:

- Changing the Appearance of a Graph (Plot or Histogram) (p. 1125)

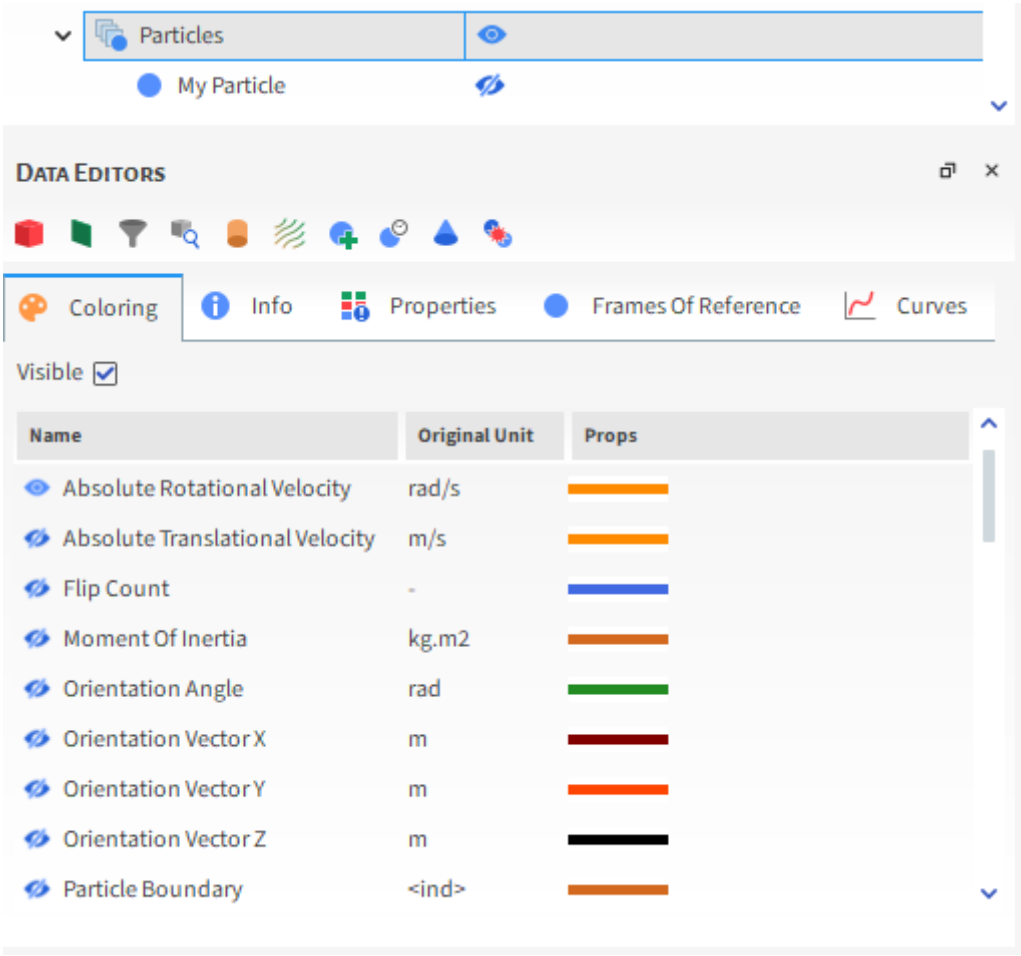
ABOUT USING THE COLORING TAB TO CHANGE A PLOT OR HISTOGRAM

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

Using the options on the Coloring tab is one way you can change what is displayed in a Plot or Histogram window.

When a Plot or Histogram window is selected, the Coloring tab for the simulation entity (p. 1395) you have selected in the Data panel shows the data sets that are already calculated and ready to display in the window (as shown below).

Figure 5.112: Coloring tab when a plot or histogram window is selected for the Particles entity



The active data sets being shown in the selected window are indicated with an "open" eye icon. (See also Show/Hide Components by Using Eye Icons and Checkboxes (p. 113)). You can add or remove data shown in the selected plot or histogram by "opening" or "closing" these eye icons accordingly.

In addition, you can use the **Props** column to change the color and display of the lines, dots, or dashes representing the data in the selected window. (See also Double-Click a Plot Line, Bar, or Point to Change Its Appearance (p. 121)).

What would you like to do?

- Use the Coloring Tab to Change a Plot or Histogram (p. 1154)

See Also:

- Changing the Appearance of a Graph (Plot or Histogram) (p. 1125)

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

USE THE COLORING TAB TO CHANGE A PLOT OR HISTOGRAM

1. From the Workspace (p. 1396) or **Windows** panel, select the Plot or Histogram window that you want to change. (See also Graph Data within Rocky by Creating a Plot or Histogram (p. 1124).)
2. From the **Data** panel, select the simulation entity (p. 1395) that contains the properties you want to display or change within the selected window.
3. From the **Data Editors** panel, select the **Coloring** tab, and then do one or both of the following:
   - Use the eye icons to change what data sets are shown. (See also Show/Hide Components by Using Eye Icons and Checkboxes (p. 113).)
   - Double-click a data set row to change the color or display of the data line, dashes, or dots shown in the selected window. (See also Double-Click a Plot Line, Bar, or Point to Change Its Appearance (p. 121).) The visualization settings you change are automatically updated in the selected window.

See Also:

- About Using the Coloring Tab to Change a Plot or Histogram (p. 1152)
- Changing the Appearance of a Graph (Plot or Histogram) (p. 1125)

5.3.3. Export Data and Images Out of Rocky

There are four main ways you can export data and images out of Rocky:

- By saving a snapshot of the selected Rocky window
- By exporting Plot or Histogram data to a separate CSV file you can review outside of Rocky
- By copying an image or data set to the clipboard
- By collecting and then exporting boundary forces for FEM analysis

Besides data and images, you may also export individual particle shapes and geometry components out of Rocky as well. You can also use Ansys Workbench to link data from a Rocky project directly to other Ansys tools, such as Mechanical or Fluent.

What would you like to do?

- Learn more About Exporting Data and Images Out of Rocky (p. 1155)
- Learn more About Collecting and then Exporting Geometry Forces Data for Later FEM Analysis (p. 1159)
- Save a Snapshot of the Rocky Window (p. 1163)
- Export Data into a CSV File (p. 1164)
- Copy an Image or Data Set to the Clipboard (p. 1164)
- Export Boundary Forces for FEM Analysis (p. 1165)

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

- Export a Particle Shape to an STL File (p. 698)
- Export a Geometry Component to an STL File (p. 520)
- Learn more About Rocky and Ansys Workbench Integration (p. 144)

See Also:

- View and Save Simulation Statistics (p. 1097)

About Exporting Data and Images Out of Rocky

There are four main ways you can export data out of Rocky:

- by saving a snapshot of the selected Rocky window
- by exporting Plot or Histogram data to a separate CSV file you can review outside of Rocky
- by copying information, including the snapshot or data values within the window you've selected, to paste into another program
- by having Rocky collect forces data for the simulation geometries and then exporting that data to a CSV file you can use in a separate FEA program. (See also About Collecting and then Exporting Geometry Forces Data for Later FEM Analysis (p. 1159).)

Notes:

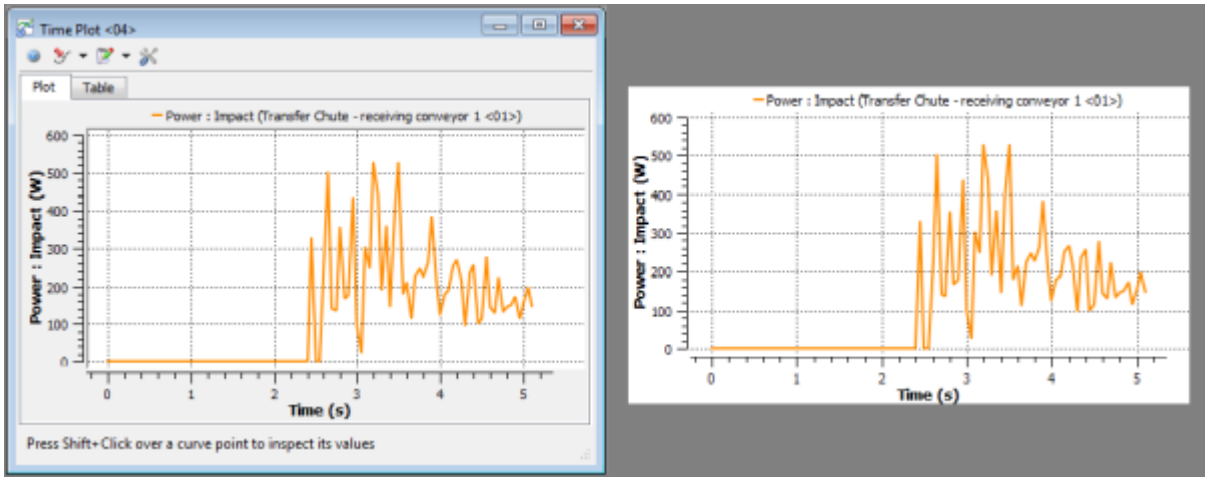
- You may also export individual particle shapes and geometry components out of Rocky as well. These are covered in separate topics. (See also Export a Particle Shape to an STL File (p. 698) and Export a Geometry Component to an STL File (p. 520).)
- You may also use Ansys Workbench to exchange Rocky data with other Ansys tools. (See also About Rocky and Ansys Workbench Integration (p. 144).)

SAVING A SNAPSHOT OF A ROCKY WINDOW

Taking an snapshot of the 3D View, Motion Preview, Particles Details, or graph (plot or histogram) saves an image of only the display area within the selected window. The other window items surrounding the display area, including buttons and on-screen instructions are not included in the final image (as shown below).

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

Figure 5.113: A Time Plot window in Rocky (left); the saved snapshot image of the same window (right)

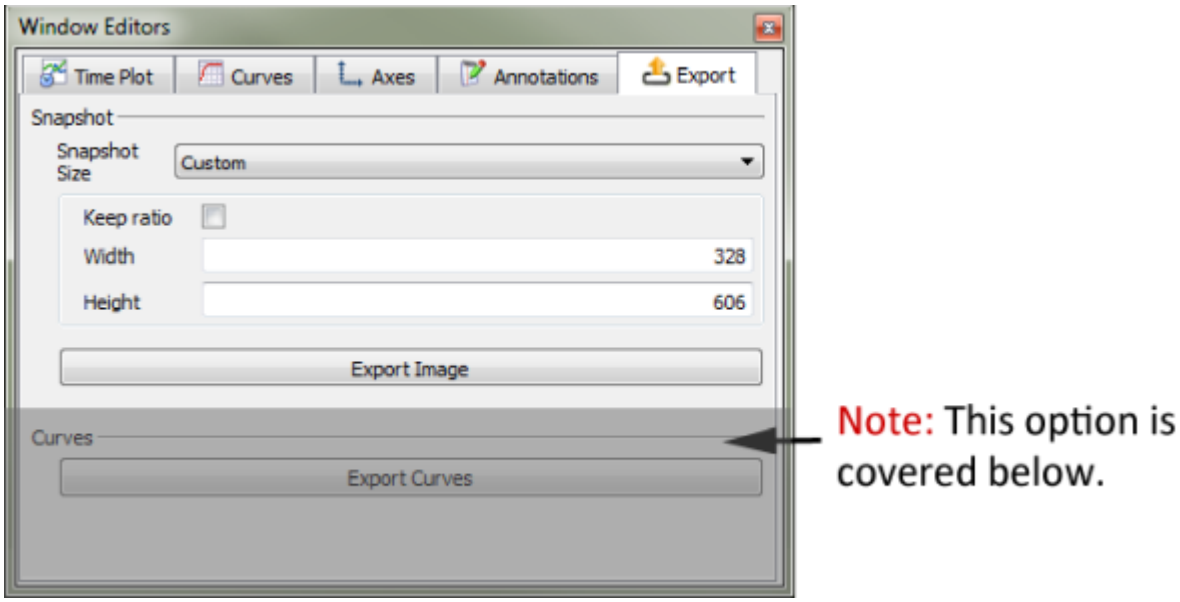


There are several ways to save an image in Rocky, which produce slightly different outcomes as described below:

- Save a Snapshot of the Rocky Window (p. 1163), which is accessible through the Export tab on the Window Editors panel and allows you to accomplish tasks like changing the dimensions of the image prior to saving it (Plots and Histograms only) or setting the magnification of the image (3D View, Motion Preview, and Particles Details windows only). Note: For Plots and Histograms, you can access this same information by right-clicking within the window area, pointing to **Export** and then selecting **Image**.
- Save an Image of the Rocky Window (p. 858), which is accessible through the Windows menu and uses only the current dimensions of the window when saving the image.

See the images and tables below to understand more about how to use the Window Editors panel to save a snapshot.

Figure 5.114: Export Image settings in the Window Editors panel (Plots and Histograms only)



Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

Figure 5.115: Export settings in the Window Editors panel (3D Views, Motion Preview, and Particles Details windows only)

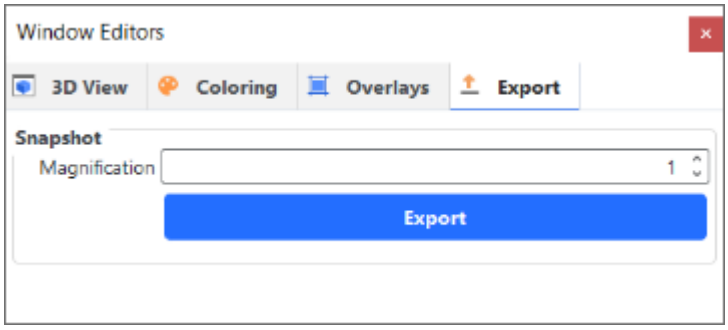


Table 1: Export Image settings for Plots and Histograms in the Window Editors panel

Setting	Description	Range
Snapshot Size	Enables you to choose the width and height of the image to be saved. Specifically: • **Window Size** uses the same dimensions as the selected window. • **Custom** enables you to set custom image dimensions.	Window Size; Custom
Keep ratio	When **Snapshot Size** is set to **Custom**, this enables you to keep the aspect ratio of your image relative to the window size. When enabled, you can change only the **Width** value and Rocky will calculate the **Height** value accordingly. When cleared, you can change both the **Width** and **Height** values.	Turns on or off
Width	When **Snapshot Size** is set to **Window Size**, this displays the width of the selected window in pixels. When **Snapshot Size** is set to **Custom**, this enables you to set a custom width value.	Positive values
Height	When **Snapshot Size** is set to **Window Size**, this displays the height of the selected window in pixels. When **Snapshot Size** is set to **Custom**, this enables you to set a custom height value.	Positive values

Table 2: Export Image settings for 3D View, Motion Preview, and Particles Details windows in the Window Editors panel

Setting	Description	Range
Magnification	Enables you to increase the level of magnification for the image you are exporting. For example, a value of 1 saves the image at the same level of magnification as the selected window. A value of 2 saves the image at double the magnification, a value three at triple the magnification, and so on.	Whole numbers 1 or greater

EXPORTING PLOT OR HISTOGRAM DATA TO A CSV FILE

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

To review your data in a CSV file outside of Rocky, you must first create a plot or histogram of the data you want to export. (See also Graphing (Plot or Histogram) a Data Set Within Rocky (p. 1101).)

After you choose to **Export Curves** on the **Export** tab of the **Window Editors** panel (Figure 5.116: Time Plot, Export Curves settings in the Window Editors panel (p. 1158)), you can select which curves you want to include in the CSV file on the **Export Configuration** dialog (Figure 5.117: Export Configuration dialog (p. 1158)).

Note:

You can access this same information by right-clicking within the window area, pointing to **Export** and then selecting **Curves**.

Figure 5.116: Time Plot, Export Curves settings in the Window Editors panel

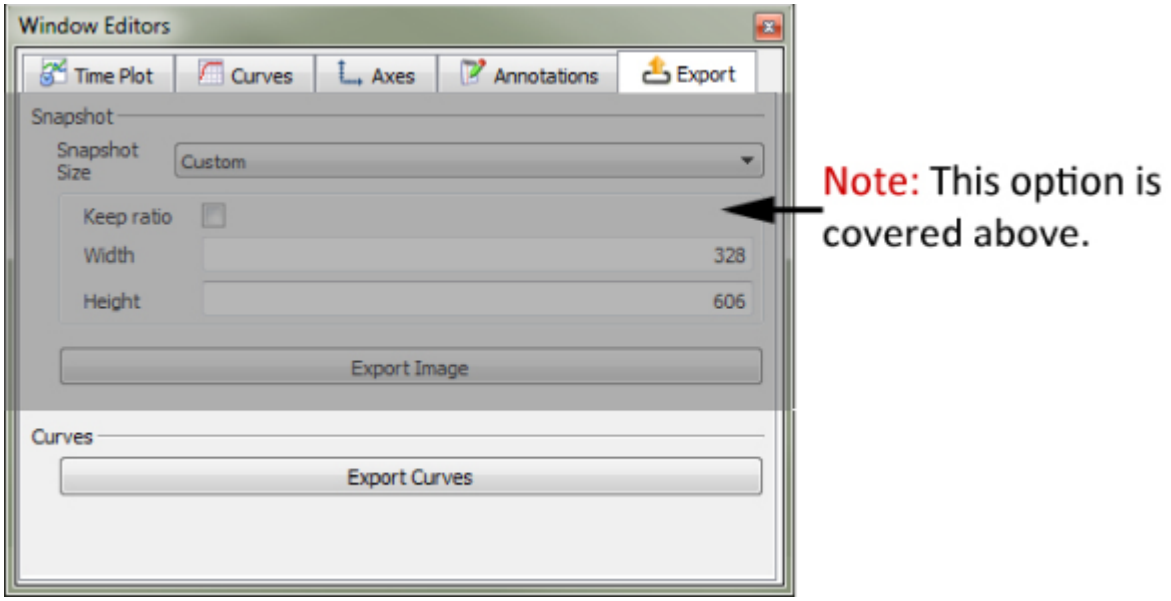
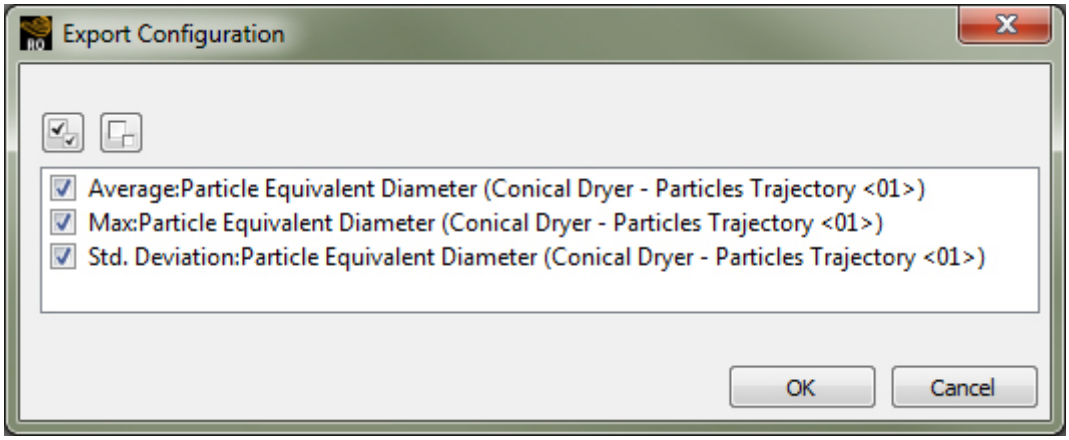


Figure 5.117: Export Configuration dialog



COPYING SNAPSHOT OR DATA FOR PASTING LATER

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

If you don't want to save a copy of your image or data, you can also choose to place a temporary copy on the clipboard and then paste it into another program later. You may accomplish this from the **Edit** | **Copy** menu, or from the right-click menu within the window area you want to copy.

Choosing **Image** copies the image using the current dimensions of the window to the clipboard, which can then be pasted into an image editor or other software. This is available for all Rocky windows.

Choosing **Values** copies the underlying data used to create the curves in a plot or histogram window to the clipboard, which can then be pasted into a spreadsheet or other software.

What would you like to do?

- Save a Snapshot of the Rocky Window (p. 1163)
- Export Data into a CSV File (p. 1164)
- Copy an Image or Data Set to the Clipboard (p. 1164)
- Learn more About Collecting and then Exporting Geometry Forces Data for Later FEM Analysis (p. 1159)
- Learn more About Rocky and Ansys Workbench Integration (p. 144)
- Export Boundary Forces for FEM Analysis (p. 1165)

See Also:

- View and Save Simulation Statistics (p. 1097)
- Export a Particle Shape to an STL File (p. 698)
- Export a Geometry Component to an STL File (p. 520)

About Collecting and then Exporting Geometry Loads Data for Later FEM Analysis

If you are interested in analyzing how the particle loading will affect your structure, then you can have Rocky record those loads and export them for use in a third-party FEA tool (p. 1391), such as Ansys Mechanical. To do so, ensure you have enabled the collection of **Forces for FEM Analysis** for Boundary Collisions Statistics before processing your simulation (see also About Collision Statistics for Boundaries (p. 312)). (Note: If you are doing this integration through Workbench, this checkbox will be automatically enabled for you.) After processing, you are then able to export the recorded data into a .csv file. This file can be used in a separate FEA tool—such as Ansys Mechanical—for analysis.

FEM loads can be exported for one or more geometry components in your simulation, and for a single time step or a time step range that you define. One CSV file is created for each of the selected time steps and includes the recorded loads acting on all selected geometries as a result of particle interaction.

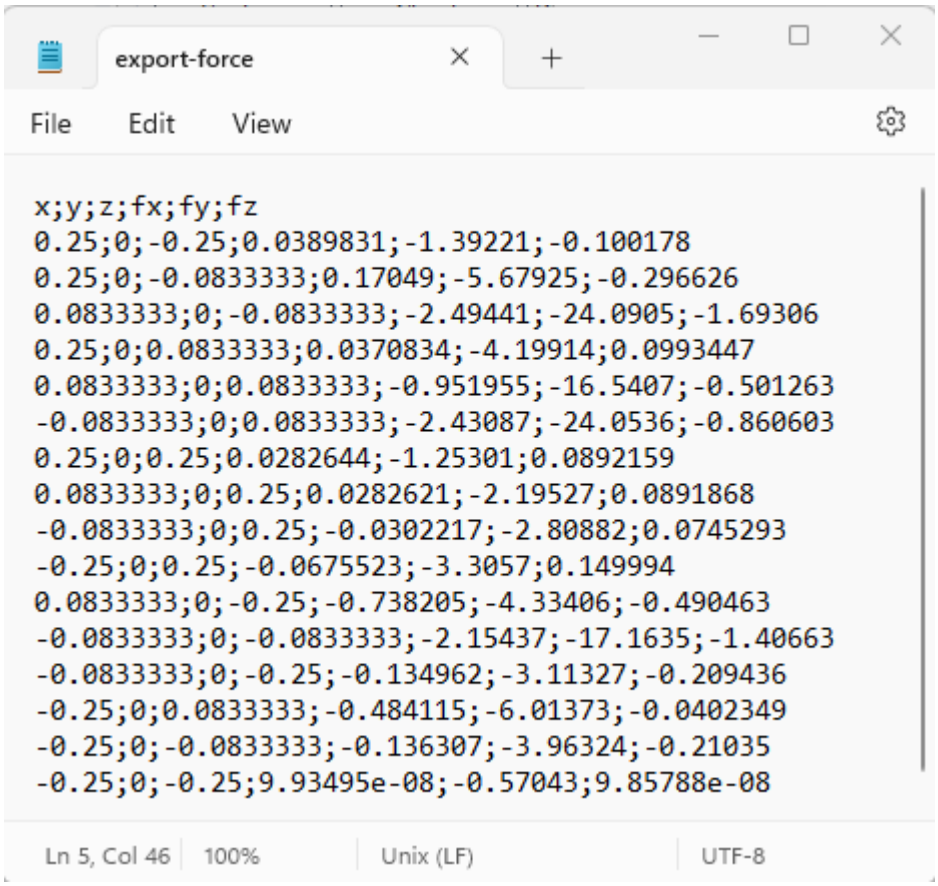
ABOUT THE EXPORTED CSV FILE

The CSV files that you export will show the values for **the selected load type** in each coordinate. The image below illustrates how the data is arranged: the first , the second and the third columns refer to the X, Y and Z coordinates from the geometries you selected, respectively. The fourth , fifth

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

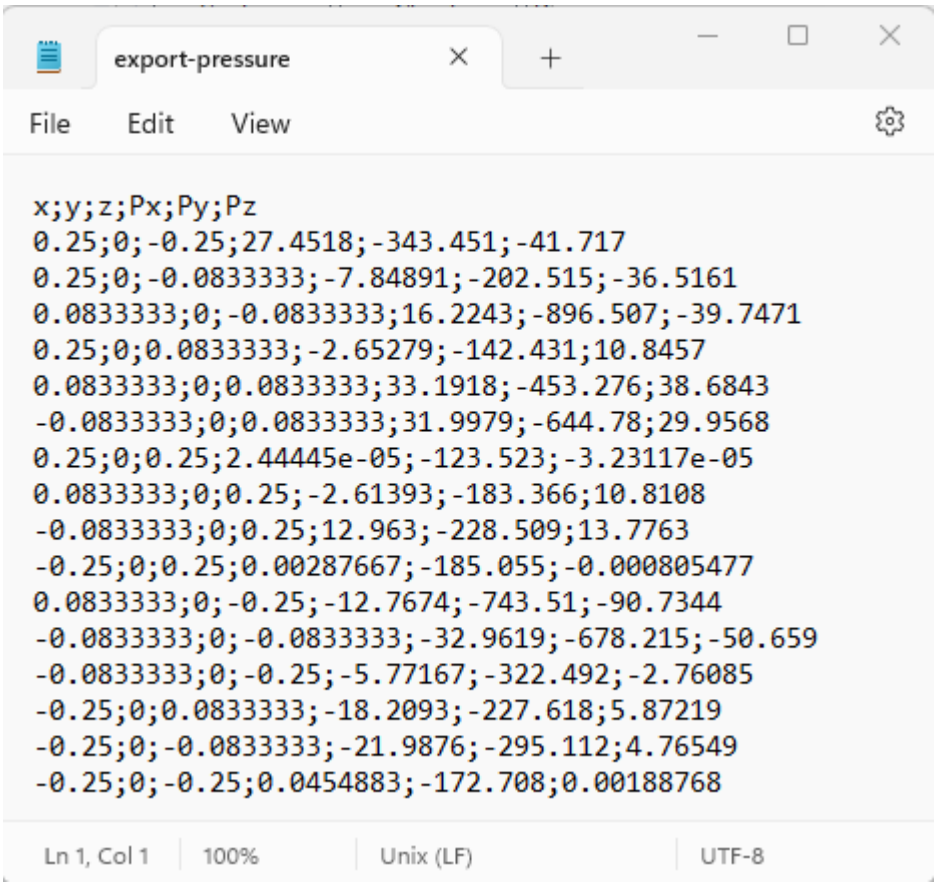
and sixth columns refer to the FEM loads components in directions X, Y and Z respectively, at each coordinate.

Figure 5.118: Example of force exported CSV file



Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

Figure 5.119: Example of pressure exported CSV file



Tip:

You can also use Ansys Workbench to share the data Rocky collects directly with Ansys Mechanical using a Static Structural Model, or a Transient Structural Model. (See also About Rocky and Ansys Workbench Integration (p. 144).)

EXPORT LOADS PARAMETER DEFINITIONS

Use the image and table below to understand how to export geometry loads into a CSV file.

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

Figure 5.120: Export Loads dialog

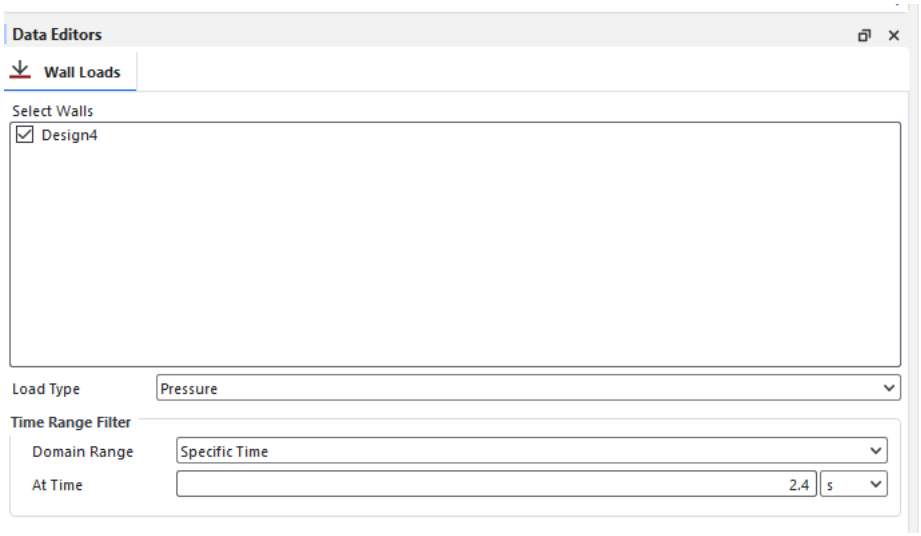


Table 1: Export Loads dialog settings

Setting	Description	Range
Select Walls	Enables you to choose which geometry data will be included in the export.	Turns on or off
Load Type	Enables you to choose what you want to include in the export, between pressure and force.	Pressure; Force.
Domain Range	Defines what time steps of the simulation are included in the export. The options are as follows: • **Application Time Filter**: Use the range settings on the Time Filter dialog to limit the time steps used. (For more information, see About the Time Toolbar.) • **All**: Use all available time steps in the simulation. • **Last Output**: Use only the very last (most recent) time step that was calculated. • **Time Range**: Limit the time step values by a specific range that you choose. (Uses the **Initial** and **Final** values you set.) • **Specific Time**: Choose only a single specific time step. (Uses the **At Time** value you set.) • **After Time**: Use all time steps available after a specific time period of your choosing. (Uses the **Initial** value you set.)	Application Time Filter; All; Last Output; Time Range; Specify Time; After Time; Time Range Relative to Simulation End.

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

Setting	Description	Range
	• **Time Range Relative to Simulation End**: Specify how long before the very end of the simulation to include time steps. (Uses the **Range from end** value you set.)	
Initial	When **Time Range** or **After Time** is chosen for **Domain Range**, this is the starting time to begin the time step selection.	Any value between 0 and the final simulation time.
Final	When **Time Range** is chosen for **Domain Range**, this is the ending time to stop the time step selection.	Any value between the **Initial** time and the final simulation time.
At Time	When **Specific Time** is chosen for **Domain Range**, this is the exact moment in which the time step will be selected.	Any value between 0 and the final simulation time.
Range from end	When **Time Range Relative to Simulation End** is chosen for **Domain Range**, this is the period of time before the final simulation time in which time steps will be included in the selection. For example, when you want to include only the last X seconds of the simulation.	Any value between 0 and the final simulation time.

What would you like to do?

- Export Boundary Forces for FEM Analysis (p. 1165)
- Learn more About Rocky and Ansys Workbench Integration (p. 144)
- Learn more About Collision Statistics for Boundaries (p. 312)

See Also:

- View and Save Simulation Statistics (p. 1097)
- Export a Particle Shape to an STL File (p. 698)
- Export a Geometry Component to an STL File (p. 520)

Save a Snapshot of the Rocky Window

1. In the Workspace (p. 1396), select the 3D View, Motion Preview, Particles Details, Plot, or Histogram window for which you want to save a snapshot.
2. From the **Window Editors** panel, select the **Export** tab, and then under **Snapshot**, enter what you want.
3. Click **Export Image**. The **Snapshot** dialog appears.

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

4. From the **Snapshot** dialog, enter a folder location, **File Name** and **Save as Type** information, and then click **Save**. The image is saved to the location you chose and Rocky asks you whether you want to open the snapshot.

Tip:

For Plots and Histograms only, you can also access this same information by right-clicking within the window area, pointing to **Export** and then selecting **Image**.

See Also:

- About Exporting Data and Images Out of Rocky (p. 1155)
- View and Save Simulation Statistics (p. 1097)

Export Data into a CSV File

1. From the **Windows** panel, select the plot or histogram containing the data that you want to export. Tip:Use the **View** menu to show any panels that might not currently be visible in the UI.
2. From the **Window Editors** panel, select the **Export** tab and then under **Curves**, click **Export Curves**.
3. From the **Export Configuration** dialog, ensure under **Export** that the check box for the data set(s) you want are selected, and then click **OK**.
4. From the **Export Curve(s)** dialog, choose a location and enter a file name for the CSV file, and then click **Save**.

Tips:

- You can perform this same process by right-clicking within the window area, pointing to **Export** and then selecting **Curves**.
- You can copy the data to the clipboard instead of saving it by right-clicking within the window area, point to **Copy** and then selecting **Curves**.

See Also:

- About Exporting Data and Images Out of Rocky (p. 1155)

Copy an Image or Data Set to the Clipboard

1. In the Workspace (p. 1396), select the 3D View, Motion Preview, Particles Details, Plot, or Histogram window for which you want to copy an image or data.
2. From the **Edit** menu, point to **Copy** and then do one of the following:
   - To copy a snapshot of the window to the clipboard, click **Image**.
   - To copy the underlying data to the clipboard, click **Values**.

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

The copied information is now available for you to paste into another program, such as a spreadsheet or image editor.

See Also:

- About Exporting Data and Images Out of Rocky (p. 1155)
- Save a Snapshot of the Rocky Window (p. 1163)
- Export Data into a CSV File (p. 1164)

Export Boundary Forces for FEM Analysis

1. Set up your simulation as you normally would.(See also Setting Up a Simulation (p. 411).) Tip:You can also use Ansys Workbench to share the data Rocky collects directly with Ansys Mechanical using a Static Structural Model, or a Transient Structural Model. (See also About Rocky and Ansys Workbench Integration (p. 144).)
2. Before processing your simulation, ensure that you have enabled the **Forces for FEM Analysis** Boundary Collision Statistics option. (See also About Collision Statistics for Boundaries (p. 312).) Note: If you are setting up your simulation within Workbench, this option will be enabled for you.
3. Process your simulation. (See also Processing a Simulation (p. 907).)
4. When processing is complete, do all of the following:
    a. From the **Data** panel under **Geometries**, right-click one of the boundaries for which you want to export forces, point to **Export**, and then click **Geometry Loads**.
    b. From the **Export Loads** dialog, select the geometries for which you want to export forces data, and then under **Time Range Filter**, define which time step or range you want included in the export. (See also About Collecting and then Exporting Geometry Forces Data for Later FEM Analysis (p. 1159).)
    c. Click **OK**.
    d. From the **Select file to export** dialog, choose a location and name for the .csv file, and then click **Save**. One .csv file is saved for each time step output in the range that you selected. These .csv files can then be used in your FEA program to analyze geometry forces.

See Also:

- About Collecting and then Exporting Geometry Forces Data for Later FEM Analysis (p. 1159)
- About Rocky and Ansys Workbench Integration (p. 144)
- About Solver Parameters (p. 464)
- About Collision Statistics for Boundaries (p. 312)

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

5.4. Create and Save an Animation

Once you have completed a simulation, you might want to save the results as an animation file so that you can view it and share it outside of Rocky.

You can use any type of view window as a basis for setting up an animation, including 3D View, Motion Preview, and Particles Details windows.

What would you like to do?

- Learn more About Creating and Saving an Animation (p. 1166)
- Set up an Animation (p. 1169)
- Use the View in the Window to Change the Qualities of a Key Frame (p. 1171)
- Use the View in the Key Frame to Edit the Qualities of that Key Frame (p. 1171)
- Edit the Settings and Key Frames Included in an Already Set-Up Animation (p. 1172)
- Save an Already Set-Up Animation as an AVI (p. 1173)
- Save a Static Image of Each Individual Frame in an Animation (p. 1173)

See Also:

- Analyzing a Simulation (p. 931)
- About 3D View Windows (p. 1040)
- About Motion Preview Windows (p. 572)
- About Particles Details Windows (p. 679)

5.4.1. About Creating and Saving an Animation

Animations are saved as AVI files, which can be opened with many types of media players, and can be set to show linear and spline transitions, 360° wraparound views, and variable frame rates.

When you create an animation, you can choose which portion of the simulation files to include, enabling you to make an animation of the whole simulation or just a portion of it.

You can create an animation based upon any one of the view window (p. 1396) types in Rocky, including 3D Views, Motion Previews, and Particles Details windows. Any visualization settings that you have set for the view window including the colors, particle shapes, labels, and view angles, will also be replicated in the animation exactly as shown in the window, so it is important to ensure that those settings are as you want before adding your Key Frames.

Important:

Animation settings are linked to a single view window. The window that is active when you start adding your Key Frames is the one linked to the Animation settings that are shown. While a single Animation set can be based on only a single window, you may have

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

as many different Animation sets as you have open view windows. Because of this relationship, however, you will also lose your Animation settings if you close the window to which the Animation is linked. If that happens, you can use the Undo function to get your settings back. (See also Undo or Redo a Single Action at Once (p. 108)).

Tips:

- For precise animations requiring exact positioning and sequencing regardless of the time step, consider using Custom Cameras when creating your Key Frames.
- For creating many similar animations of different simulations, consider using Rocky's script tool to record a repeatable set of UI tasks. (For more information, see Creating and Using Your Scripts (p. 296)).
- To see a walk-through example of creating an animation, refer to the following Workshop:

Tutorial - Transfer Chute in the *Rocky Tutorial Guide*

Use the figure and table below to understand the various parameters you can set for your animation file.

Figure 5.121: Animation panel

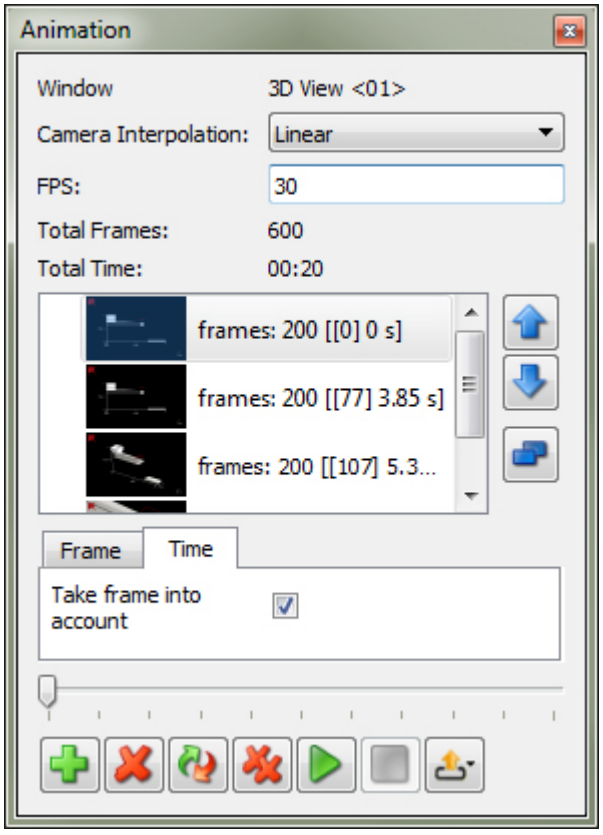


Table 1: Animation Options

Setting	Description	Range
Window	Displays the name of the selected Rocky window to which the current Animation settings apply.	Automatic value
Camera Interpolation	Determines the type of transition between Key Frames: • **Linear** produces straight-line transitions between Key Frames. • **Spline** produces curved motion transitions between Key Frames. • **No Interpolation** results in a jumping effect between Key Frames with no transition.	Linear; Spline; No Interpolation
FPS	Sets how many Frames Per Second (FPS) the animation will include. A higher FPS will produce a shorter, faster video with higher-quality playback. A lower FPS will produce a longer, slower video with lower-quality playback.	Positive value (20 is recommended for most animations)
Total Frames	Shows how many frames will be included in the overall animation. This number is automatically calculated based upon the Key Frames selected, the **FPS**, and the **Number of Frames** values that are set.	value
Total Time	Shows the length of the animation in real time. This number is automatically calculated based upon the Key Frames selected, the **FPS**, and the **Number of Frames** values that are set.	Automatic value
Take frame into account	When this check box is selected, the Key Frame will show during the animation the progressive simulation results for the time frame until the next Key Frame. When the check box is cleared, the selected Key Frame will show only a static image of the simulation results until the next Key Frame.	Turns on or off
Number of Frames	Sets how many individual frames this Key Frame will be separated into. These are the number of frames used before the next Key Frame is activated. The minimum number of frames that can be set is 2. **Tip:** If you want your animation to reflect real time, it is important to keep your frame rate consistent. A general rule of thumb is to set the Number of Frames value equal to the number of seconds between Key Frames multiplied by the FPS you have set. #Frames = #s x #FPS **Example:** If you set your Key Frames to start at 0s, 10s, 15s, and 20s respectively, and you want a FPS of 10 and a real time video of 20s, then:	2 or more

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

Setting	Description	Range
	• The Number of Frames for the first Key Frame should be set to 100 (10s x 10FPS = 100 Frames). • The Number of Frames for the second and third Key Frames should be set to 50 (5s x 10FPS = 50 Frames). • The Number of Frames for the fourth (last) Key Frame does not affect the animation.	

What do you want to do?

- Set up an Animation (p. 1169)
- Use the View in the Window to Change the Qualities of a Key Frame (p. 1171)
- Use the View in the Key Frame to Edit the Qualities of that Key Frame (p. 1171)
- Edit the Settings and Key Frames Included in an Already Set-Up Animation (p. 1172)
- Save an Already Set-Up Animation as an AVI (p. 1173)
- Save a Static Image of Each Individual Frame in an Animation (p. 1173)

See Also:

- Analyzing a Simulation (p. 931)
- About 3D View Windows (p. 1040)
- About Motion Preview Windows (p. 572)
- About Particles Details Windows (p. 679)

5.4.2. Set up an Animation

1. Open the simulation containing the results you want to animate.
2. Ensure the **Animation** panel is visible. (From the **Tools** menu, click **Animation**.)
3. Create a Key Frames list by doing the following:
   a. Find or create a view window (p. 1396) (3D View, Motion Preview, or Particles Details) with the visualization settings you want to animate, including the rotation and level of zoom. Note: This window will be linked to the animation settings that follow.
   b. Using the time step dropdown and/or arrows in the toolbar, choose the particular time step with which you want your animation to start. (For example, if you want your animation to start at the beginning, choose the **[0] 0 s** time step.)
   c. From the bottom of the **Animation** panel, click the **Add Key Frame** button. An image of the time step and view you created shows in the Key Frames list.

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

d. Set up your next Key Frame by doing one of the following:

- To show the same view for a period of time, move only your time step forward as desired but don't change the view.
- To transition to another view over a period of time, move your time step forward and change your view as desired.

e. From the bottom of the **Animation** panel, click the **Add Key Frame** button. An image of the new time step and view you created shows in the Key Frames list.

f. Repeat steps 3d-3e as many times as necessary to complete your animation set up. Tip:Use the buttons at the bottom and side of the **Animation** panel to help you delete, copy, move, and update Key Frames that you've already created.

4. From the top of the **Animation** panel, determine what kind of transitions you want between your Key Frames by choosing an option from the **Camera Interpolation** list. (See also About Creating and Saving an Animation (p. 1166).)

5. Determine the speed, length, and quality of your animation by setting the values you want in the **FPS** field for the animation, and **Number of Frames** field for each Key Frame. Use the values displayed for **Total Frames** and **Total Time** to guide you in your selections.

6. At the bottom of the **Animation** panel, click the **Play** button to view a preview of your animation in the Rocky Workspace, and then edit your Key Frames and animation options as desired.

Transition Tips

- To transition quickly between views, leave no time step gap between Key Frames.
- To transition slowly between views, leave a time step gap between Key Frames equal to the desired transition length.
- To create a 360° wraparound view, include five Key Frames—the first and last with the same exact view, the third at a point halfway around the desired rotation path, and the second and forth views at opposite points in between. For a smooth transition, ensure equal time step gap settings between views. **For example**, create Key Frames at the following camera presets and time steps: Z [5s], X [6s], -Z [7s], -X [8s], Z [9s].

See Also:

- About Creating and Saving an Animation (p. 1166)
- Use the View in the Window to Change the Qualities of a Key Frame (p. 1171)
- Use the View in the Key Frame to Edit the Qualities of that Key Frame (p. 1171)
- Edit the Settings and Key Frames Included in an Already Set-Up Animation (p. 1172)
- Save an Already Set-Up Animation as an AVI (p. 1173)
- Save a Static Image of Each Individual Frame in an Animation (p. 1173)

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

5.4.3. Use the View in the Window to Change the Qualities of a Key Frame

1. Change what is shown in the view window (p. 1396) (3D View, Motion Preview, or Particles Details) by doing any or all of the following:
   - From the view window upon which the animation is based, modify the visualization settings as desired, including the rotation and level of zoom.
   - From the Time toolbar, use the dropdown menu and/or arrows to change the time step as desired. (See also About the Time Toolbar (p. 1279).)
   - From the **Animation** panel, change the **Number of frames** value as desired.
2. From the **Animation** panel, select the Key Frame to which you want to apply the changes.
3. At the bottom of the **Animation** panel, click the **Update Current Key Frame** button. The image of the Key Frame and its settings are updated to match the 3D window accordingly.

See Also:

- About Creating and Saving an Animation (p. 1166)
- Set up an Animation (p. 1169)
- Use the View in the Key Frame to Edit the Qualities of that Key Frame (p. 1171)
- Edit the Settings and Key Frames Included in an Already Set-Up Animation (p. 1172)
- Save an Already Set-Up Animation as an AVI (p. 1173)
- Save a Static Image of Each Individual Frame in an Animation (p. 1173)

5.4.4. Use the View in the Key Frame to Edit the Qualities of that Key Frame

1. From the **Animation** panel, double-click the Key Frame you want to edit. The Key Frame view and time step location is displayed in the view window (p. 1396) (3D View, Motion Preview, or Particles Details) upon which the animation is based.
2. Do any or all of the following:
   - From the view window, modify the visualization settings as desired, including the rotation and level of zoom.
   - From the Time toolbar, use the dropdown menu and/or arrows to change the time step as desired. (See also About the Time Toolbar (p. 1279).)
   - From the **Animation** panel, change the **Number of frames** value as desired.
3. At the bottom of the **Animation** panel, click the **Update Current Key Frame** button. The image of the Key Frame and its settings are updated accordingly.

See Also:

- About Creating and Saving an Animation (p. 1166)

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

- Set up an Animation (p. 1169)
- Use the View in the 3D View Window to Change the Qualities of a Key Frame (p. 1171)
- Edit the Settings and Key Frames Included in an Already Set-Up Animation (p. 1172)
- Save an Already Set-Up Animation as an AVI (p. 1173)
- Save a Static Image of Each Individual Frame in an Animation (p. 1173)

5.4.5. Edit the Settings and Key Frames Included in an Already Set-Up Animation

1. From the Workspace, select the view window (p. 1396) (3D View, Motion Preview, or Particles Details) upon which the animation you want to change is based. The Animation panel displays the Key Frames and settings for the selected window.
2. Do any or all of the following:
   - Change the **Camera Interpolation** and/or **FPS** values as desired.
   - Change the order of a Key Frame by first selecting it, and then using the buttons on the side of the **Animation** panel to move it up or down in the list.
   - Make a copy of an existing Key Frame by first selecting it, and then on the side of the **Animation** panel, clicking the **Duplicate Key Frame** button.
   - Remove a Key Frame by first selecting it, and then at the bottom of the **Animation** panel, clicking the **Remove Key Frame** button.
   - Delete all Key Frames in the animation by clicking the **Remove All Key Frames** button at the bottom of the **Animation** panel.
   - Create a new Key Frame by clicking the **Add Key Frame** button at the bottom of the **Animation** panel.

See Also:

- About Creating and Saving an Animation (p. 1166)
- Set up an Animation (p. 1169)
- Use the View in the Window to Change the Qualities of a Key Frame (p. 1171)
- Use the View in the Key Frame to Edit the Qualities of that Key Frame (p. 1171)
- Save an Already Set-Up Animation as an AVI (p. 1173)
- Save a Static Image of Each Individual Frame in an Animation (p. 1173)

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

5.4.6. Save an Already Set-Up Animation as an AVI

Tip:

Before saving your AVI file, ensure that you have the video compression and quality settings you want specified in the **Options** | **Preferences** dialog. For more information, see About Setting Global Preferences (p. 227).

1. From the bottom of the **Animation** panel, click the **Export Animation** button, and then click **AVI**.
2. From the **Export Animation to AVI** dialog, choose the location to which you want to save the AVI file.
3. In the **File name** box, enter a name for the file, and then click **Save**. The **Animation Export** screen appears to show you the status of your save and a preview of the progress. When the save is complete, the **Animation Export** screen disappears and Rocky asks you if you want to open the AVI file you just saved.

See Also:

- About Creating and Saving an Animation (p. 1166)
- Set up an Animation (p. 1169)
- Use the View in the 3D View Window to Change the Qualities of a Key Frame (p. 1171)
- Use the View in the Key Frame to Edit the Qualities of that Key Frame (p. 1171)
- Edit the Settings and Key Frames Included in an Already Set-Up Animation (p. 1172)
- Save a Static Image of Each Individual Frame in an Animation (p. 1173)

5.4.7. Save a Static Image of Each Individual Frame in an Animation

Tip:

Before you start, review the **Total Frames** value displayed in the **Animation** panel to ensure you have set up your animation the way you want. The number of images exported will equal this value.

1. From the bottom of the **Animation** panel, click the **Export Animation** button, and then click **Images**.
2. Form the **Export Animation to Images** dialog, choose the location to which you want to save the series of images.
3. In the **File name** box, enter a name for the images and then from the **Save as type** list, choose what image format you want.

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

4. Click **Save**. The **Animation Export** screen appears to show you the status of your save and a preview of the progress. When the save is complete, the **Animation Export** screen disappears. Each image file name will be appended with the corresponding frame number.

See Also:

- About Creating and Saving an Animation (p. 1166)
- Set up an Animation (p. 1169)
- Use the View in the 3D View Window to Change the Qualities of a Key Frame (p. 1171)
- Use the View in the Key Frame to Edit the Qualities of that Key Frame (p. 1171)
- Edit the Settings and Key Frames Included in an Already Set-Up Animation (p. 1172)
- Save an Already Set-Up Animation as an AVI (p. 1173)

5.5. Filter Views and Data with User Processes

User Processes are the methods by which Rocky enables you to limit—or perhaps focus—the view you want to see and the data you want to analyze. Through the various shapes, planes, and settings, you can divide, reduce, or filter the simulation area to better control the results you see and data you graph. Cut planes and analysis boxes are two examples of User Processes that you can create in Rocky.

5.5.1. Creating a User Process

You create a User Process by first selecting the entity (p. 1395) you want to filter (e.g., individual geometry component, Particles, Contacts, 1-Way LBM, 1-Way Fluent, or another User Process) and then choosing the kind of process—Cube, Plane, Filter, Particles Trajectory, Cylinder, Polyhedron (Envelope), and so on—you want to use.

The new User Process that is created is considered to be linked to the original entity and will display only the chosen subset of information. (See also See Input/Output Relationships in the Data Panel When Component Names are Bold (p. 114).)

You can also duplicate a User Process; analyze an individual triangle, block, or node by creating a Cell Inspector User Process; and create User Processes from other User Processes.

If after you set up your User Processes, you want to apply them to another project, you can also choose to export and reuse them in other projects. (See also Reuse 3D View Window, Workspace, and User Process Setups in Other Projects (p. 403).)

5.5.2. About Particles User Processes

User Processes created from Particles can work differently or have extra functionality when compared to User Processes created from other kinds of entities. For example, with a User Process created from Particles, you may do any of the following:

- Limit Particles by a particular time range by creating a Particles Time Selection User Process.

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

- From a Cube, Cylinder, or Custom (Polyhedron) shaped User Process created from the Particles entity, apply Eulerian Statistics, which divides the shape into individual blocks which themselves can be analyzed.
- Apply Particles Calculations to any Particles selection (Cube, Cylinder, or Polyhedron) to have certain functions such as Flip Count and Residence time apply to particles when in the selected area but retain their values when they leave it.
- From a Cube, Cylinder, or other User Process created from the Particles entity, convert that sub-set of Particles into its associated Contacts by creating a Particle to Contact User Process.

5.5.3. About Particle <-> Contact User Processes

Particle to Contact and **Contact to Particle** User Processes—collectively referred to as *Particle* <-> *Contact* User Processes—are unique User Processes designed specifically to convert a sub-set of Particles to their respective Contacts (and vice versa).

What do you want to do?

- Analyze Data within a Defined Shape (p. 1175)
- Analyze Data with a Clip or Cut Plane (p. 1195)
- Analyze Data by Property Type (p. 1202)
- Analyze the Particles Trajectory (p. 1208)
- Inspect an Individual Cell (p. 1211)
- Duplicate a User Process (p. 1219)
- Export User Process Geometry Triangles to an STL File (p. 1220)
- Analyze Particles within a Time Selection (p. 1221)
- Divide a Cube or Cylinder User Process into Many Analysis Blocks (p. 1226)
- Retain Particle Data Calculated within a User Process by Using Particle Calculations (p. 1242)
- Reuse 3D View Window, Workspace, and User Process Setups in Other Projects (p. 403)
- Analyze Contacts from a Sub-Set of Particles (p. 1271)
- Analyze Particles from a Sub-Set of Contacts (p. 1274)

See Also:

- Analyzing a Simulation (p. 931)

5.5.4. Analyze Data within a Defined Shape

When you analyze data within a defined shape, you determine what subset of the simulation area you want to focus on. This area is based on the size and location of the User Processes shape you choose, as follows:

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

- **Cube** enables you to create a box-like shape.
- **Cylinder** enables you to create a full or partial cylindrical shape, with or without a center hole.
- **Polyhedron (Envelope)** enables you to import a custom shape that you've created outside of Rocky and have saved as an .stl file.

Data calculated within any of these User Process shapes can be retained outside of those selections by using Particle Calculations (see also About Particles Calculations (p. 1242)) but only the Cube and Cylinder User Processes shapes can have Eulerian Statistics applied to them (see also About Eulerian Statistics (p. 1226)).

What would you like to do?

- Learn more About the Cube User Process (p. 1176)
- Learn more About the Cylinder User Process (p. 1181)
- Learn more About the Polyhedron (Envelope) User Process (p. 1186)
- Analyze Data within a Cube (p. 1193)
- Analyze Data within a Cylinder (p. 1194)
- Analyze Data within a Custom Shape by using the Polyhedron (Envelope) User Process (p. 1194)

About the Cube User Process

The Cube User Process enables you to define within the simulation boundaries a three dimensional analysis box of whatever length, width, and height you want. In this way, you can limit the data you analyze to only the selection contained by the box. This method is useful for measuring the total mass of particles within a rock box, for example, or for counting how many times particles flip at the bottom of a rotating mill.

This kind of User Process can be applied to individual geometry components, Particles, 1-Way LBM, 1-Way Fluent, or any other User Process. See the image and table below to learn more about the Cube User Process.

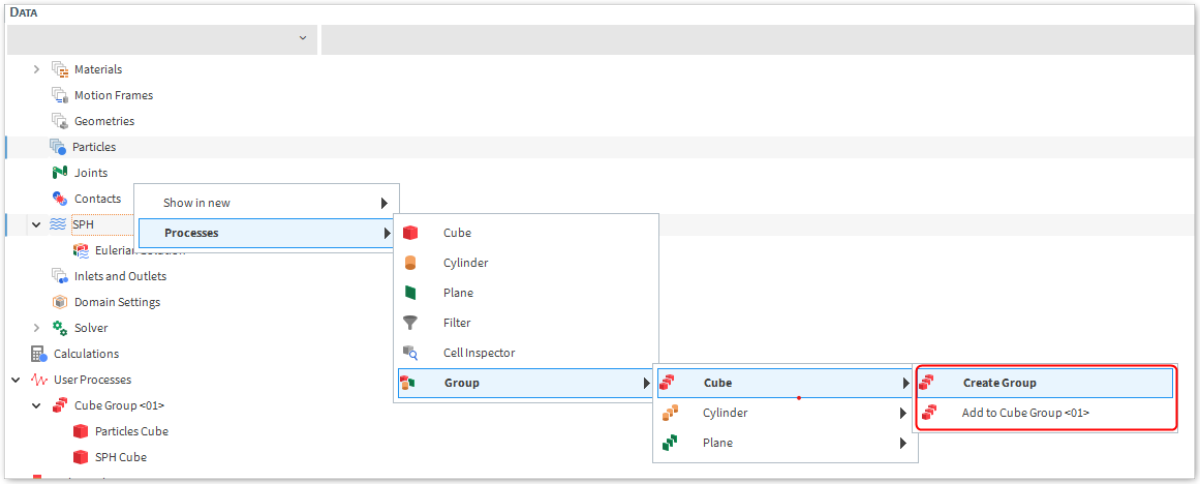
Important:

The **Cube** User Process shape supports application to multiple entities within a single simulation, enabling extended statistical analysis across several domains simultaneously.

To create a **Cube Group**, multi-select the entities you want to be part of the group, right-click them and select Processes | Group | Cube | **Create Group** (or **Add to Cube Group** to include selections in an existing group). All Cubes within a group inherit the same configurations from their group.

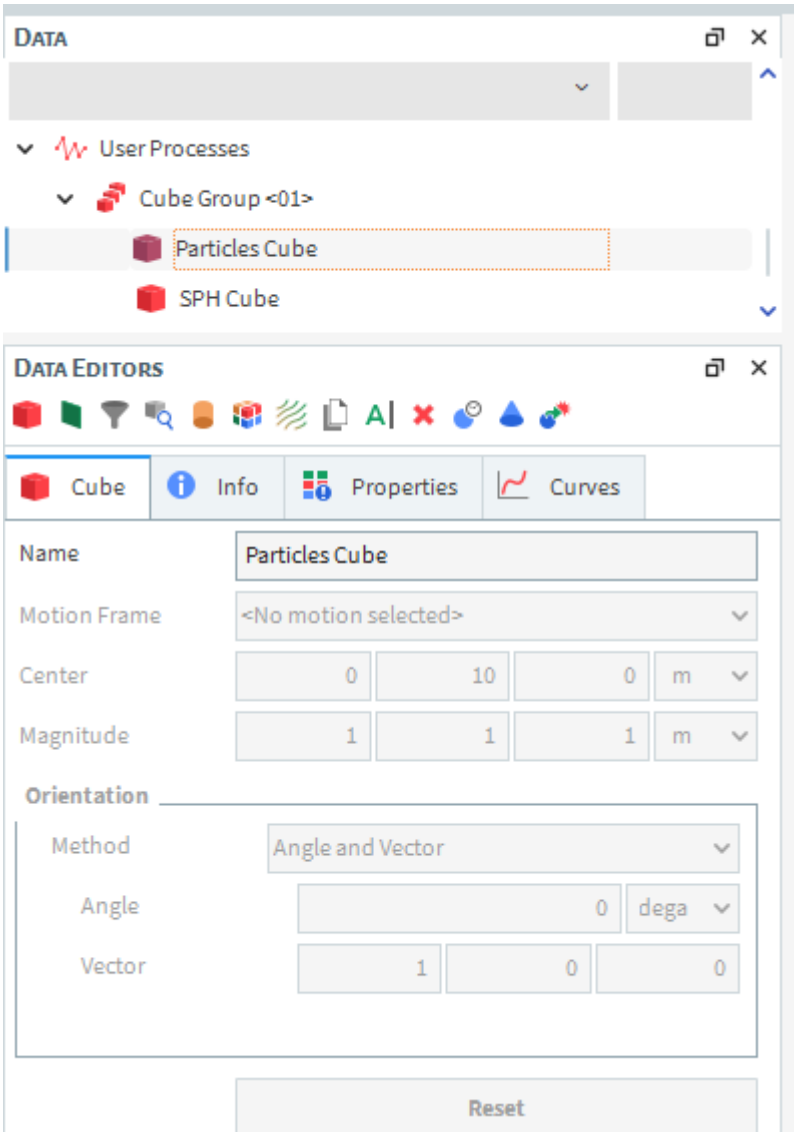
Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

Figure 5.122: How to create a Cube Group (in the image: from Particles and SPH)



Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

Figure 5.123: Example of Cube Group (each Cube inherits Motion Frame, Center, Magnitude and Orientation from their group - Cube Group <01>, in the example).



Tip:

To see walk-through examples of simulations that make use of the Cube User Process, refer to the following Tutorials:

- Tutorial - Transfer Chute in the *Rocky Tutorial Guide*
- Tutorial - Vibrating Screen in the *Rocky Tutorial Guide*

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

- Tutorial - High Pressure Grinding Roll (HPGR) in the *Rocky Tutorial Guide*

Tip:

You can Use Colored Directional Handles (p. 126) to change the shape and location.

CUBE USER PROCESS PARAMETER DEFINITIONS

Figure 5.124: Cube User Process applied to the Particles entity

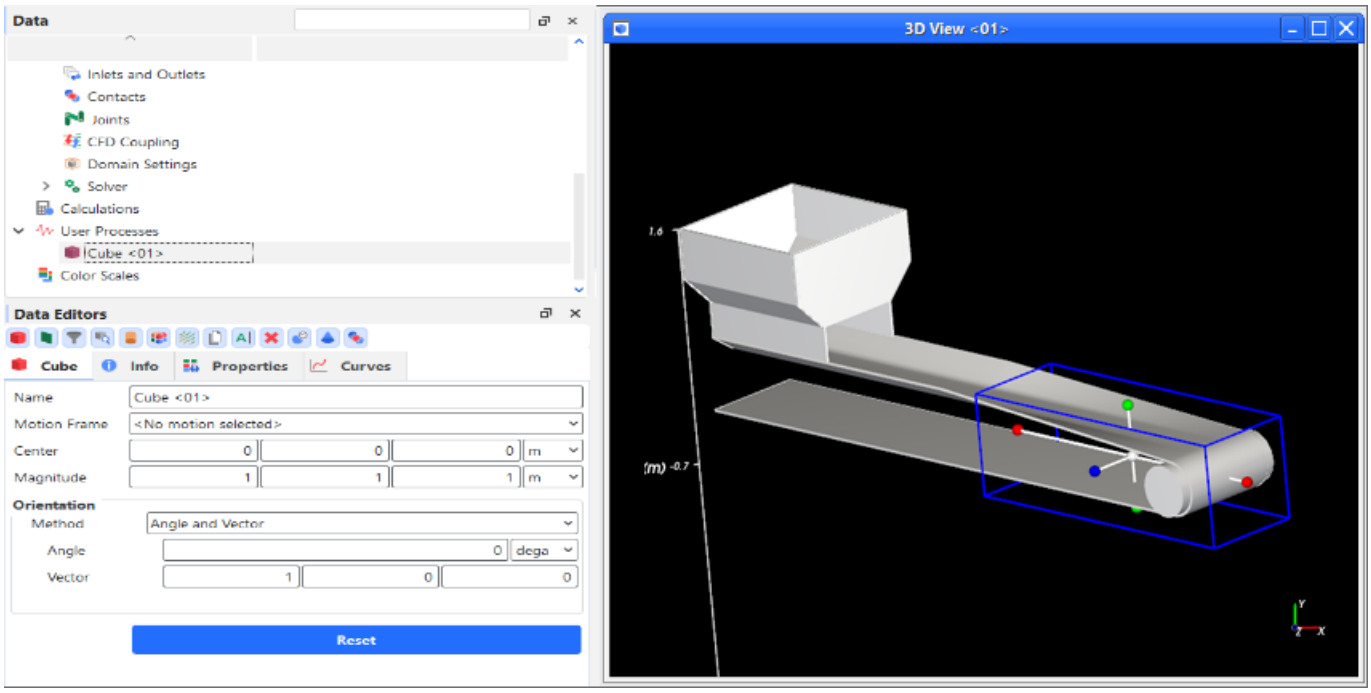


Table 1: Cube User Process parameter options

Setting	Description	Range
Name	Enables you to specify a unique identifier for the User Process. **Note:** This same name will appear in the **Data** panel under **User Processes**.	No limit
Motion Frame	Enables you to select which Motion Frame you want assigned to the geometry. (See also About Creating and Applying Motion Frames (p. 533).)	Automatically provided
Center	The X, Y, and Z coordinate locations of the box's center point.	Any value
Magnitude	The length of the box's three sides as specified in the X, Y, and Z directions.	Any value

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

Setting	Description	Range
Orientation Method	Enables you to select how you want to define the orientation of the Cube shape. Specifically: • **Angles** enables you to define angles of rotation in three directions, the order of which you can also specify. • **Angle and Vector** enables you define one vector and one angle of rotation around it. • **Basis Vectors** enables you to define the X, Y, and Z directions of the Cube's local basis.	Angles; Angle and Vector; Basis Vectors
Method Angles		
Order	When **Angles** is selected for **Orientation**, this defines the order in which the three **Rotation** text fields will be applied.	XYZ; ZXY; YXZ; YZX; ZXY; ZYX
Local Angles	When **Angles** is selected for **Orientation**, this defines what coordinate system will be used as a basis for the angle specified. Specifically: • When enabled, the angle will be based on the local coordinate system. • When cleared, the angle will be based on the global coordinate system.	Turns on or off
Rotation	When **Angles** is selected for **Orientation**, this is the degree of box rotation in each of the three directions specified by the **Order** provided.	Any value
Method Angle and Vector		
Angle	When **Angle and Vector** is selected for **Orientation**, this is the angle the box will rotate around the **Vector** defined.	Any value
Vector	When **Angle and Vector** is selected for **Orientation**, this is the X, Y, and Z components that define the vector around which the box will rotate, using the **Angle** defined.	No limit but the values entered will be normalized
Method Basis Vectors		
X direction	When **Basis Vectors** is selected for **Orientation**, this is the coordinate values defining the first of three	No limit but the values entered

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

Setting	Description	Range
	directional vectors that together define the final orientation of the box. **Tip:** To ensure correct results, make sure you define this vector as orthogonal (perpendicular) to the other two vectors.	will be normalized
Y direction	When **Basis Vectors** is selected for **Orientation**, this is the coordinate values defining the second of three directional vectors that together define the final orientation of the box. **Tip:** To ensure correct results, make sure you define this vector as orthogonal (perpendicular) to the other two vectors.	No limit but the values entered will be normalized
Z direction	When **Basis Vectors** is selected for **Orientation**, this is the coordinate values defining the third of three directional vectors that together define the final orientation of the box. **Tip:** To ensure correct results, make sure you define this vector as orthogonal (perpendicular) to the other two vectors.	No limit but the values entered will be normalized

See Also:

- Analyze Data within a Cube (p. 1193)

About the Cylinder User Process

The Cylinder User Process enables you to define within the simulation boundaries a three dimensional analysis tube, arch, or slice of whatever diameter and height you want, and with or without a center hole. (See Figure 5.129: Cylinder shape examples: arch (left), slice (center), and tube (right) (p. 1185) below for examples.) In this way, you can limit the data you analyze to only the selection contained by the shape you determine. The method is particularly useful for analyzing certain areas of equipment with round or elliptical components, such as mills and pipes.

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

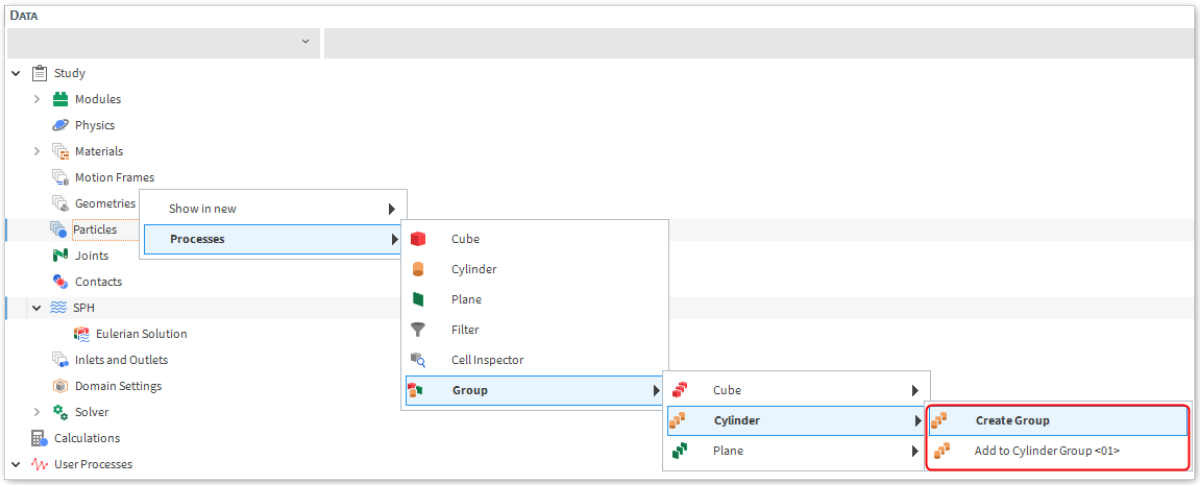
See the images and table below to learn more about the Cylinder User Process.

Important:

The **Cylinder** User Process shape supports application to multiple entities within a single simulation, enabling extended statistical analysis across several domains simultaneously.

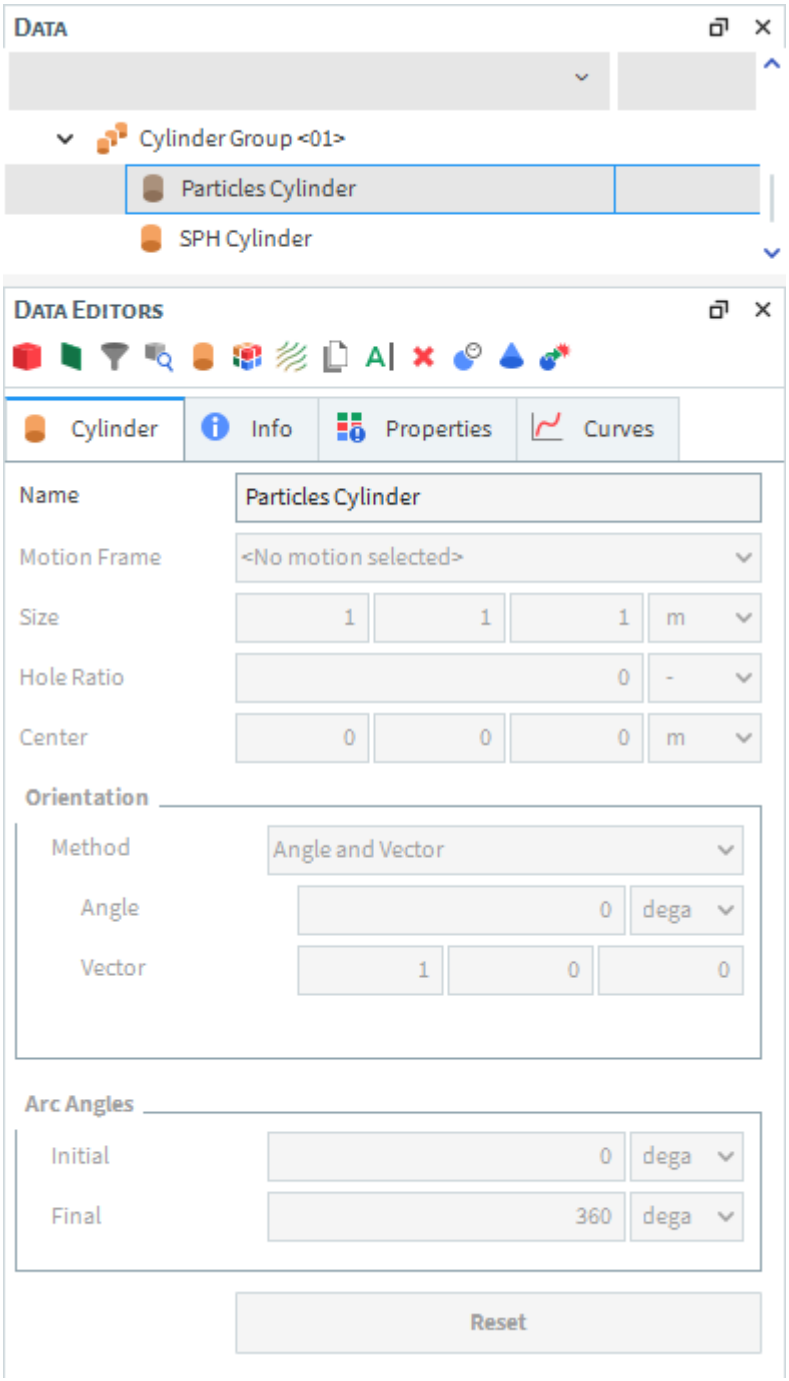
To create a **Cylinder Group**, multi-select the entities you want to be part of the group, right-click them and select Processes | Group | Cylinder | **Create Group** (or **Add to Cylinder Group** to include selections in an existing group). All Cubes within a group inherit the same configurations from their group.

Figure 5.125: How to create a Cylinder Group (in the image: from Particles and SPH)



Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

Figure 5.126: Example of Cylinder Group (each Cylinder inherits Motion Frame, Center, Magnitude and Orientation from their group - Cylinder Group <01>, in the example).



Tip:

To see walk-through examples of simulations making use of the Cylinder User Process, refer to the following Tutorials:

- Tutorial - SAG Mill in the *Rocky Tutorial Guide*

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

- Tutorial - Cone Crusher in the *Rocky Tutorial Guide*
- Tutorial - Tablet Coater in the *Rocky Tutorial Guide*
- Tutorial - Ribbon Blender in the *Rocky Tutorial Guide*

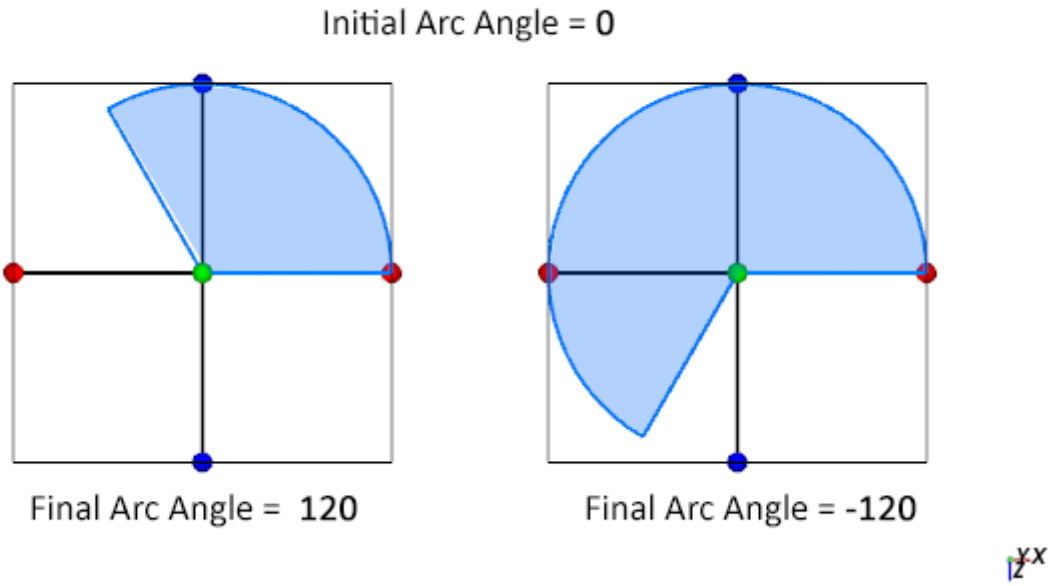
Tip:

You can Use Colored Directional Handles (p. 126) to change the shape and location.

ABOUT CYLINDER ARC ANGLES

What you set for your **Initial** and **Final** Arc Angle values determine how far around the Y axis the arc for the cylinder will be drawn. Cylinder Arc Angles follow the right-hand rule: when the Y axis points outside the screen (from the **Fit** menu, select **Camera Preset: +Y**), no matter whether you enter a positive or negative value for the **Final** parameter, the arc will be drawn from the **Initial** value counterclockwise to your **Final** value (see the image below for an illustration):

Figure 5.127: Cylinder arc results with different Final angle values



CYLINDER USER PROCESS PARAMETER DEFINITIONS

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

Figure 5.128: Cylinder User Process applied to the Particles entity

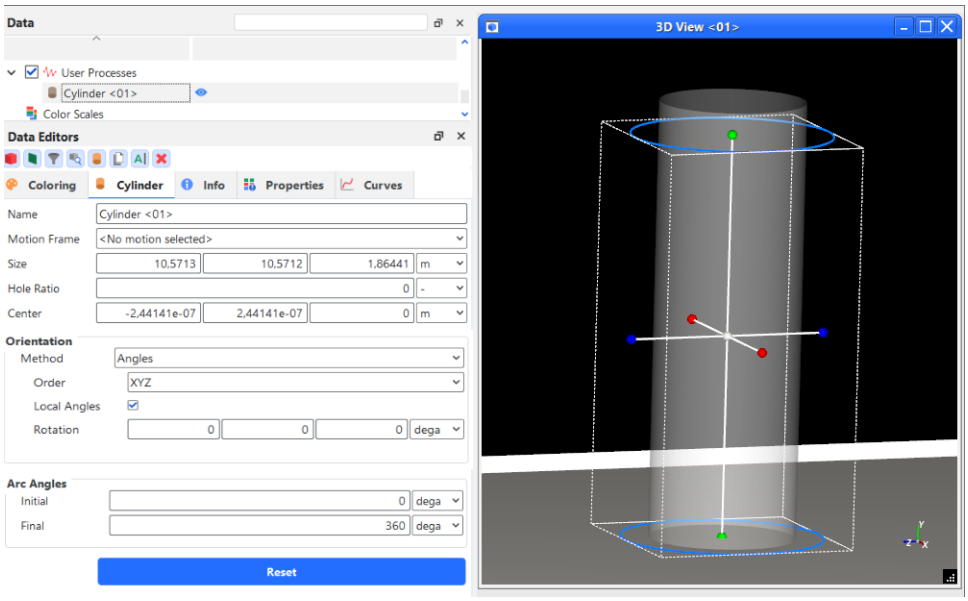


Figure 5.129: Cylinder shape examples: arch (left), slice (center), and tube (right)

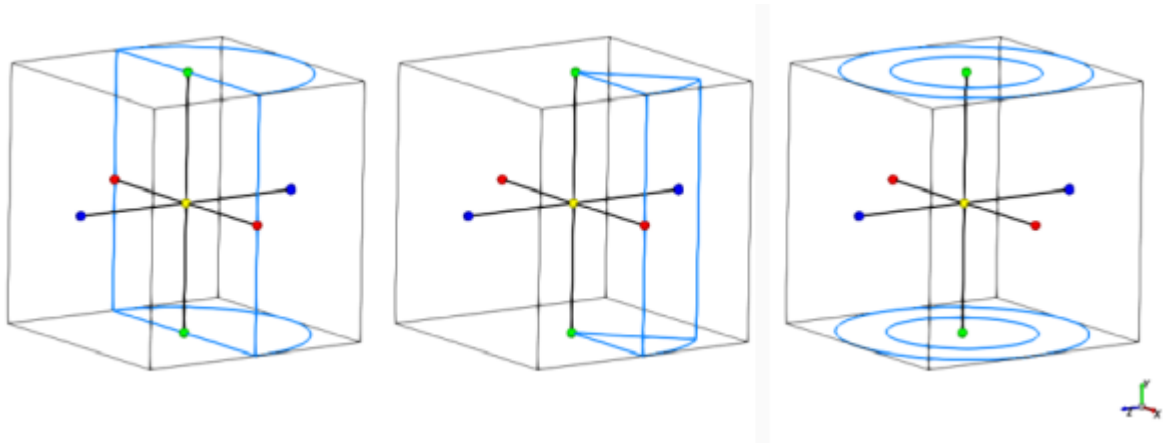


Table 1: Cylinder User Process parameter options

Setting	Description	Range
Name	Enables you to specify a unique identifier for the User Process. **Note:** This same name will appear in the **Data** panel under **User Processes**.	No limit
Motion Frame	Enables you to select which Motion Frame you want assigned to the geometry.	Automatically provided
Size	The length of the shape's three dimensions as specified in the X (diameter), Y (height), and Z (diameter) directions.	Any value

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

Setting	Description	Range
Hole Ratio	The percentage of volume that the center hole through the cylinder will take up.	0-1
Center	The X, Y, and Z coordinate locations of the shape's center point.	Any value
Method	Enables you to select how you want to define the orientation of the Cylinder shape. Specifically: • **Angles** enables you to define angles of rotation in three directions, the order of which you can also specify. • **Angle and Vector** enables you define one vector and one angle of rotation around it. • **Basis Vectors** enables you to define the X, Y, and Z directions of the Cylinder's local basis.	Angles, Angle and Vector, Basis Vectors
Arc Angles		
Initial	The initial degree point around the Y axis that the circular portion of the cylinder will be drawn. When the Y axis points outside the screen, the resulting arc will be drawn from the initial angle counterclockwise to the final angle. **Tip:** To draw a full cylinder, keep this value zero (0) and set **Final** to 360.	Any value
Final	The final degree point around the Y axis that the circular portion of the cylinder will be drawn. when the Y axis points outside the screen, the resulting arc will be drawn from the initial angle counterclockwise to the final angle. **Tip:** To draw a full cylinder, set this value to 360 and keep **Initial** at zero (0).	Any value

See Also:

- Analyze Data within a Cylinder (p. 1194)
- Use Colored Directional Handles (p. 126)

About the Polyhedron (Envelope) User Process

The Polyhedron (Envelope) User Process enables you import a custom .stl shape that is then used to define a three dimensional analysis space within the simulation boundaries. In this way, you can

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

limit the data you analyze to only the selection contained by the custom shape you import, and you can ensure that the shape you choose exactly matches the area upon which you want to focus.

This kind of User Process can be applied to Particles, SPH and geometry components and to other User Processes created from Particles, SPH or geometry components.

Tip:

To see a walk-through example that uses a custom-shaped user process area, review Tutorial - Tablet Coater in the *Rocky Tutorial Guide*.

See the images and table below to learn more about the Polyhedron (Envelope) User Process.

Figure 5.130: Import File Info dialog

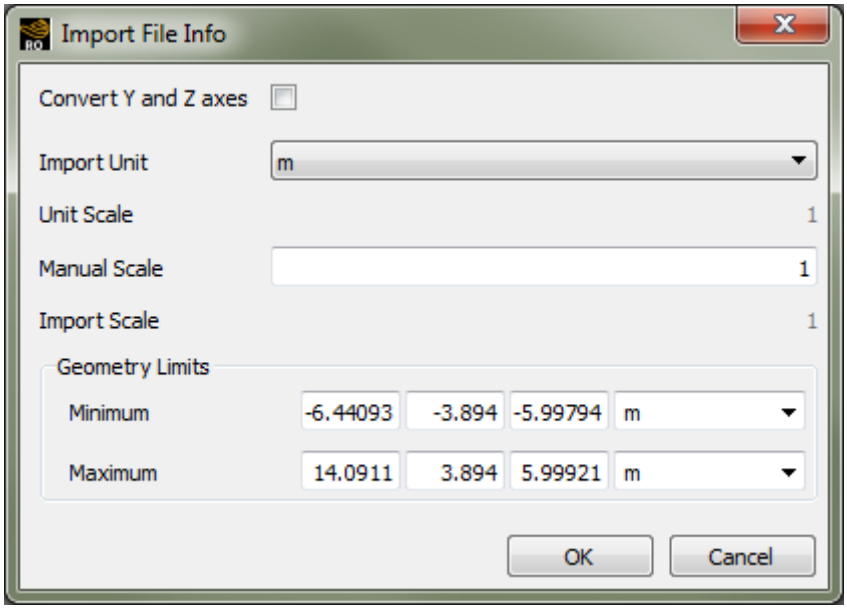


Table 1: Options displayed on the Import File Info dialog

Setting	Description	Range
Convert Y and Z axes	Selecting this option will change the axis of the imported user process shape.	Turns on or off
Import Unit	Enables you to change the units of the imported user process shape.	Various units of length
Unit Scale	Displays the unit scale based upon the **Import Unit** set. For example, if **Import Unit** is left as the default value, the Unit Scale will be 1.	Automatically determined
Manual Scale	Enables you to manually adjust the scale by any factor you want. Leaving the value at 1 will have no additional affect upon the scale.	All values
Import Scale	Displays the final import scale based upon the **Manual Scale** and **Import Unit** values set. For	Automatically determined

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

Setting	Description	Range
	example, if both those options are left as the default values, the Import Scale will be 1.	
Geometry Limits		
Minimum	The coordinates (in X Y Z format) of the lowest points geometry triangles are drawn.	No limit
Maximum	The coordinates (in X Y Z format) of the highest points geometry triangles are drawn.	No limit

Figure 5.131: Polyhedron (Envelope) User Process showing an imported almond shape

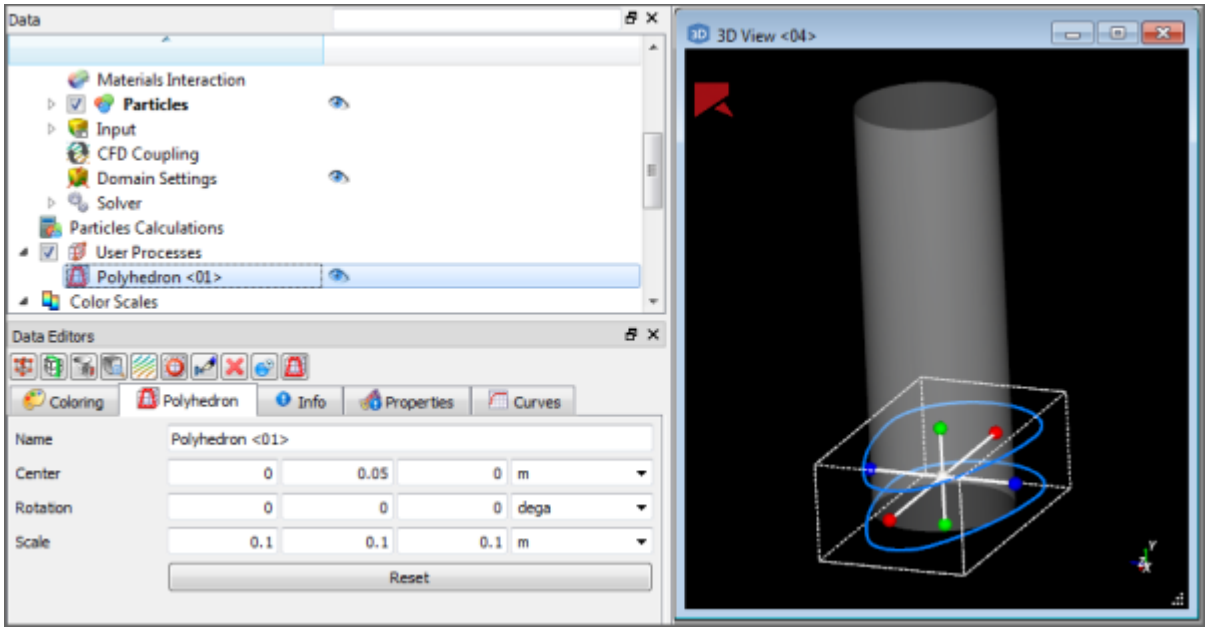


Table 2: Polyhedron (Envelope) User Process parameter options

Setting	Description	Range
Name	Enables you to specify a unique identifier for the User Process. **Note:** This same name will appear in the **Data** panel under **User Processes**.	No limit
Center	The X, Y, and Z coordinate locations of the shape's center point.	Any value
Rotation	The degree of shape rotation in each of the X, Y, and Z directions respectively.	Any value
Scale	With the original (imported) scale of the shape represented by 1, 1, 1, this is the method by which you make the shape smaller or larger in any or all of the X, Y, and Z directions respectively.	Any value

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

Setting	Description	Range
	Examples: To reduce the shape by half, change all three Scale values to 0.5. To increase only the height of the shape by 50%, change the second (Y) Scale value to 1.5.	

See Also:

- Analyze Data within a Custom Shape by using the Polyhedron (Envelope) User Process (p. 1194)

About the Surface User Process

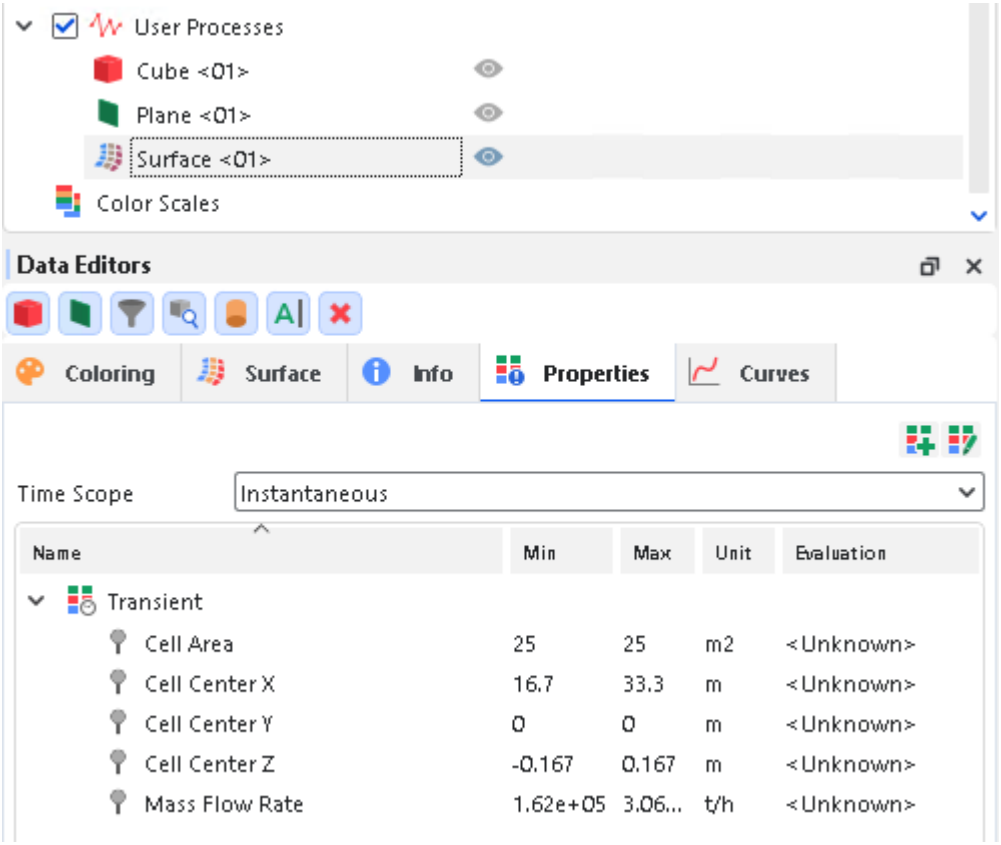
The Surface User Process enables you to import a custom .stl surface, which then can be used to measure SPH mass flow rate in post-processing phase.

Using the Surface User Process, you can measure the flow passed through each triangle of the surface, which gives you the Flow Rate and the Flow Distribution over the surface.

Its function is similar to SPH Mass Flow Rate Module, but it's important to remember that User Process is based upon calculations that were already done, making it possible to work with the relevant data collected after the simulation.

Use the figure below to see some information provided by Surface User Process:

Figure 5.132: Surface User Process Properties



SURFACE USER PROCESS PARAMETER DEFINITIONS

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

Use the image and table below to learn more about the Surface User Process.

Figure 5.133: Surface User Process with 3D tool for manipulation

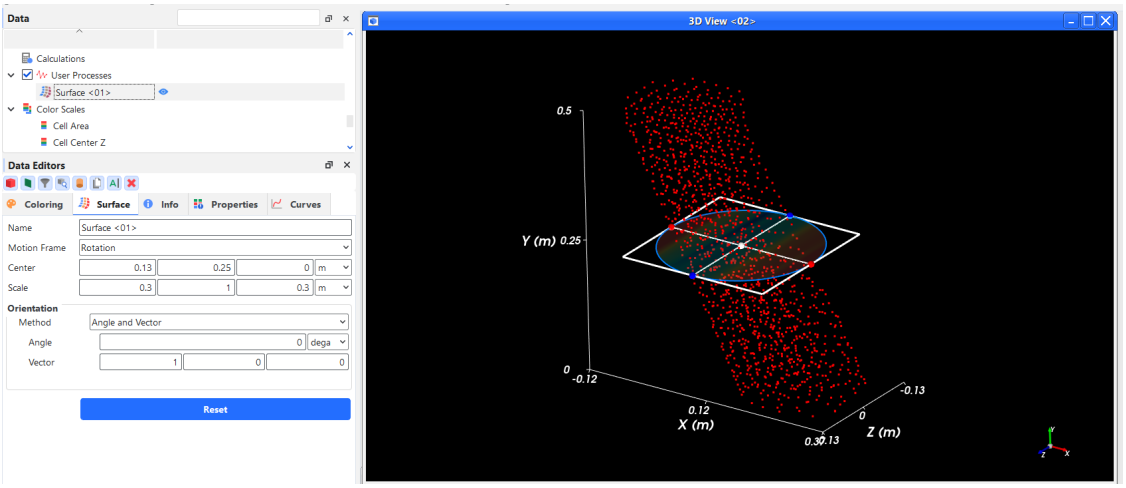


Table 5.1: Surface User Process Parameter Options

Setting	Description	Range
Name	Enables you to specify a unique identifier for the User Process. **Note:** Note: This same name will appear in the **Data** panel under **User Processes**.	No limit
Motion Frame	Enables you to select which Motion Frame you want assigned to the geometry.	Automatically provided
Center	The X, Y, and Z coordinate locations of the box's center point.	Any value
Scale	With the original (imported) scale of the shape represented by 1, 1, 1, this is the method by which you make the shape smaller or or larger in any or all of the X, Y, and Z directions respectively. Examples: To reduce the shape by half, change all three Scale values to 0.5. To increase only the height of the shape by 50%, change the second (Y) Scale value to 1.5.	Any value
Orientation		
Method	Enables you to select how you want to define the orientation of the Surface shape. Specifically: • **Angles** enables you to define angles of rotation in three directions, the order of which you can also specify. • **Angle and Vector** enables you define one vector and one angle of rotation around it.	Angles;Angle and Vector;Basis Vectors

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

Setting	Description	Range
	• **Basis Vectors** enables you to define the X, Y, and Z directions of the Cube's local basis.	
Method Angles		
Order	When **Angles** is selected for **Orientation**, this defines the order in which the three **Rotation** text fields will be applied.	XYZ; ZXY; YXZ; YZX; ZXY; ZYX.
Local Angles	When **Angles** is selected for **Orientation**, this defines what coordinate system will be used as a basis for the angle specified. Specifically: • When enabled, the angle will be based on the local coordinate system. • When cleared, the angle will be based on the global coordinate system.	Turns on or off
Rotation	When **Angles** is selected for **Orientation**, this is the degree of box rotation in each of the three directions specified by the **Order** provided.	Any value
Method Angle and Vector		
Angle	When **Angle and Vector** is selected for **Orientation**, this is the angle the box will rotate around the **Vector** defined.	Any value
Vector	When **Angle and Vector** is selected for **Orientation**, this is the X, Y, and Z components that define the vector around which the box will rotate, using the **Angle** defined.	No limit but the values entered will be normalized
MethodBasis Vectors		
X direction	When **Basis Vectors** is selected for **Orientation**, this is the coordinate values defining the first of three directional vectors that together define the final orientation of the box. Tip: To ensure correct results, make sure you define this vector as orthogonal (perpendicular) to the other two vectors.	No limit but the values entered will be normalized

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

Setting	Description	Range
Y direction	When **Basis Vectors** is selected for **Orientation**, this is the coordinate values defining the second of three directional vectors that together define the final orientation of the box. Tip: To ensure correct results, make sure you define this vector as orthogonal (perpendicular) to the other two vectors.	No limit but the values entered will be normalized
Z direction	When **Basis Vectors** is selected for **Orientation**, this is the coordinate values defining the third of three directional vectors that together define the final orientation of the box. Tip: To ensure correct results, make sure you define this vector as orthogonal (perpendicular) to the other two vectors.	No limit but the values entered will be normalized

ABOUT THE STREAMLINE USER PROCESS

Streamlines are commonly used to visualize the flow patterns or trajectories of fluid particles in a simulation, identifying regions of recirculation and assessing the overall fluid flow characteristics.

The concept of streamlines in SPH is similar to that in traditional fluid dynamics. Streamlines represent the paths that individual fluid particles would follow in the flow field. They are defined as a series of connected points or curves that are tangent to the instantaneous velocity vector of the fluid particles at each point along the path. The property uses surfaces (pre/post) to create a regular grid of source points with spacing defined by the user.

The streamlines User Process can be created on the Eulerian Solution and on its child user processes.

Note:

Streamlines can not be created from a Plane with **Exact Mode** or a Property with **Cut Mode**.

STREAMLINE USER PROCESS PARAMETER DEFINITIONS

Use the image and table below to learn more about the Streamline User Process.

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

Figure 5.134: Streamline User Process applied

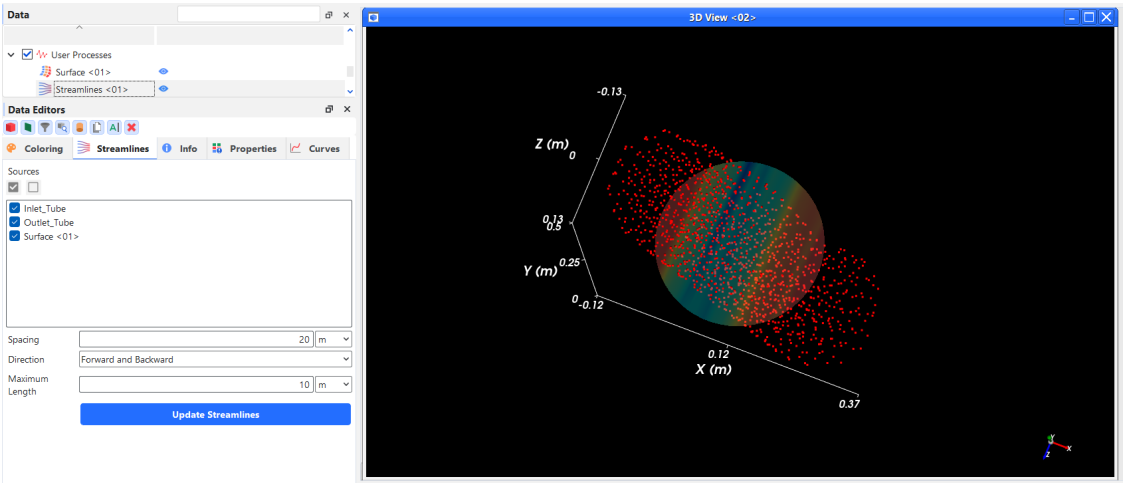


Table 5.2: Streamlines parameter options in Data Editors

Setting	Description	Range
Sources	A list of all possible sources with checkboxes, to plot streamlines from multiple origins at the same time.	Possible sources:Surface User Process;Circular Surface;Rectangular Surface;Imported Surfaces.
Spacing	It gives the spacing between the seed points on the regular grid.	Must be greater than zero.
Direction	It is possible to emit streamlines in the direction of the normal, of the surface, or in the opposite direction, or both backward and forward	Forward;Backward;Forward and Backward.
Maximum length	Defines the maximum length of the streamlines, the algorithm that calculates the streamlines would it will translate that into time to execute the calculations.	Must be greater than zero.
Update Streamlines	This button updates the streamlines, after your settings.	None.

Analyze Data within a Cube

1. From the **Data** panel, right-click the entity (p. 1395) for which you want to analyze data, point to **Processes**, and then choose **Cube**. A new Cube entry appears under **User Processes** in the **Data** panel.

2. From the **Data Editors** panel, on the **Cube** tab, enter the options you want.

Tips:

- In the 3D View window (see also Create and Modify a 3D View (p. 1040)), you can also move and resize the User Process area by clicking and dragging the handles (colored dots) representing the center, and the local X, Y, and Z locations respectively. (See also Use Colored Directional Handles (p. 126).)

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

- To revert the options back to the default settings, click the **Reset** button.

See Also:

- About the Cube User Process (p. 1176)
- Analyze Data within a Defined Shape (p. 1175)
- Use Colored Directional Handles (p. 126)

Analyze Data within a Cylinder

1. From the **Data** panel, right-click **Particles** (or a User Process created from Particles), point to **Processes**, and then choose **Cylinder**. A new Cylinder entry appears under **User Processes** in the **Data** panel.
2. From the **Data Editors** panel, on the **Cylinder** tab, enter the options you want.

Tip:

In the 3D View window (see also Create and Modify a 3D View (p. 1040)), you can also move and resize the User Process area by clicking and dragging the handles (colored dots) representing the center, and the local X, Y, and Z locations respectively. (See also Use Colored Directional Handles (p. 126).)

See Also:

- About the Cylinder User Process (p. 1181)
- Analyze Data within a Defined Shape (p. 1175)
- Use Colored Directional Handles (p. 126)

Analyze Data within a Custom Shape by using the Polyhedron (Envelope) User Process

1. Ensure that the shape you want to use is saved as an .stl file.
2. From the **Data** panel, right-click **Particles**, **SPH** or a **geometry component** (or a User Process created from Particles, SPH or a geometry component), point to **Processes**, and then choose **Polyhedron (Envelope)**.
3. From the **Select the STL file for the polyhedron** dialog, locate and select the .stl file you want, and then click **Open**.
4. From the **Import File Info** dialog that appears, choose the import options you want, and then click **OK**. A new Polyhedron entry appears under **User Processes** in the **Data** panel.

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

5. From the **Data Editors** panel, on the **Polyhedron** tab, enter the options you want.

Tip:

In the 3D View window (see also Create and Modify a 3D View (p. 1040)), you can also move and resize the User Process area by clicking and dragging the handles (colored dots) representing the center, and the local X, Y, and Z locations respectively. (See also Use Colored Directional Handles (p. 126).)

See Also:

- About the Polyhedron (Envelope) User Process (p. 1186)
- Analyze Data within a Defined Shape (p. 1175)
- Use Colored Directional Handles (p. 126)

5.5.5. Analyze Data with a Clip or Cut Plane

When you analyze data within a clip or cut plane, you gain an interior view of your simulation components. A clip plane selects all data behind the plane; a cut plane selects the data directly around the plane. Both kinds of selections are accomplished by using the Plane User Process.

What would you like to do?

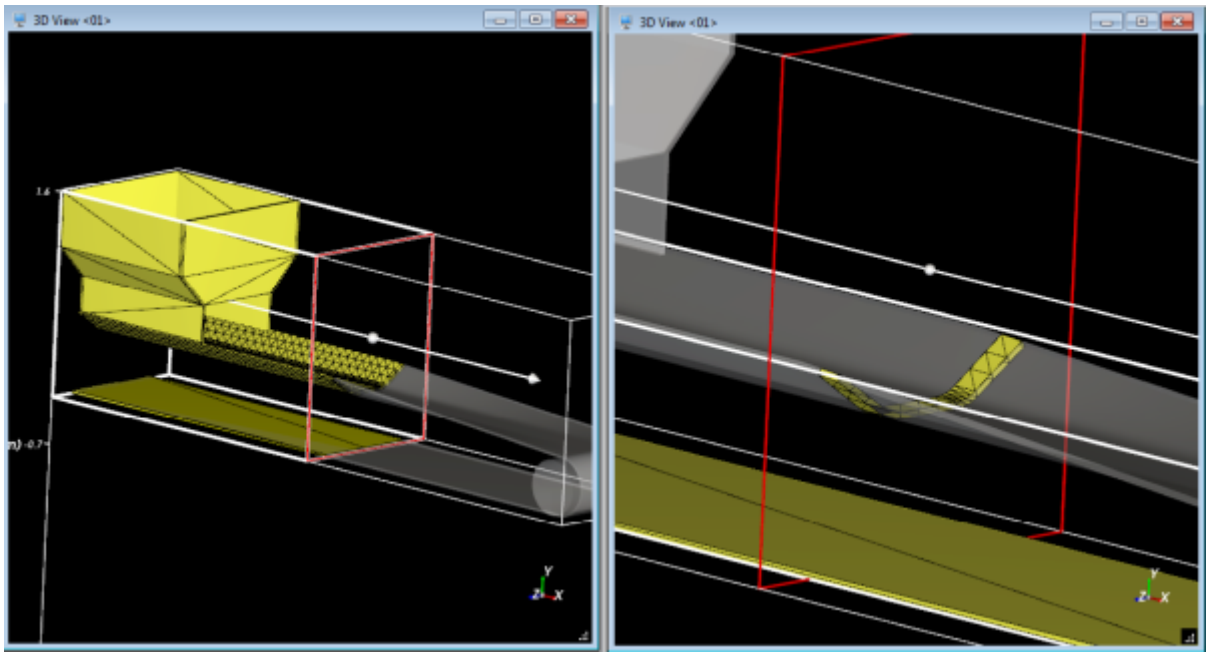
- Learn more About the Plane User Process (p. 1195)
- Analyze Data with a Clip Plane (p. 1201)
- Analyze Data with a Cut Plane (p. 1202)

About the Plane User Process

The Plane User Process enables you to limit the analysis area of your simulation based upon a plane that you define. You may either view the all results behind the plane (clip), or can limit the results to only the data immediately surrounding the plane (cut):

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

Figure 5.135: Clip Plane (left) and Cut Plane (right) examples as applied to a Feed Conveyor. Note that because Select Mode was chosen for the Cut Plane (right side of figure above), the two triangles making up the return conveyor are also included in the selection.



This kind of User Process can be applied to any entity (p. 1395) but will have different options depending upon what kind of entity you choose. Clip Planes apply to all entities while Cut Planes apply only to individual geometry components, 1-Way LBM, Eulerian Statistics, and other User Processes based on those types of entities.

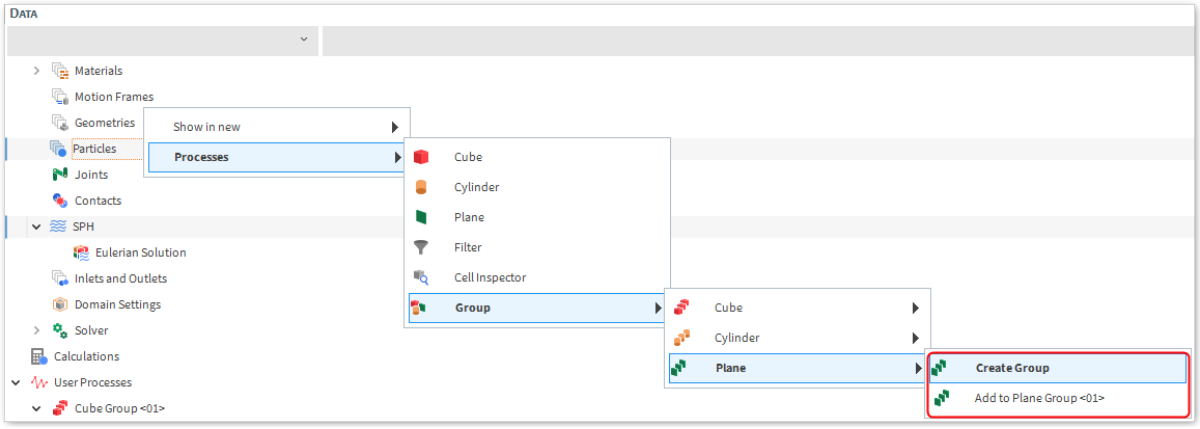
Important:

The **Plane** User Process shape supports application to multiple entities within a single simulation, enabling extended statistical analysis across several domains simultaneously.

To create a **Plane Group**, multi-select the entities you want to be part of the group, right-click them and select Processes | Group | Plane | **Create Group** (or **Add to Plane Group** to include selections in an existing group). All Planes within a group inherit the same configurations from their group.

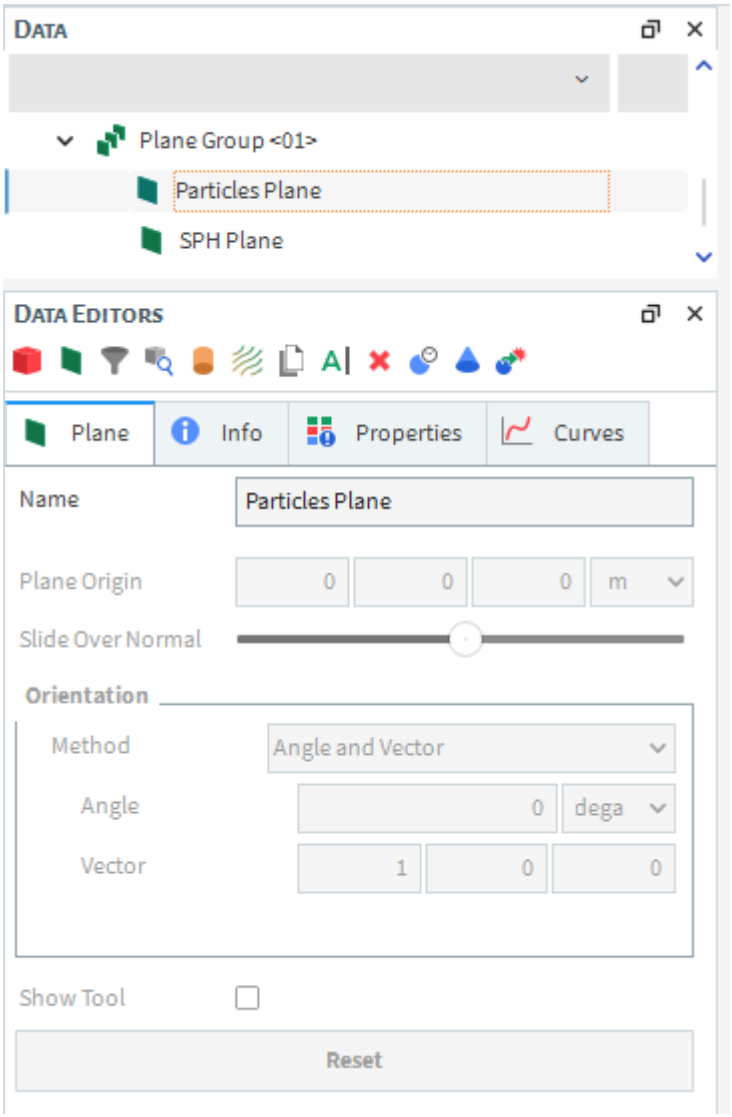
Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

Figure 5.136: How to create a Plane Group (in the image: from Particles and SPH)



Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

Figure 5.137: Example of Plane Group (each Plane inherits Motion Frame, Center, Magnitude and Orientation from their group - Plane Group <01>, in the example).



Tip:

To see walk-through examples using the Plane User Process, review the following Tutorials:

- Tutorial - Transfer Chute in the *Rocky Tutorial Guide*
- Tutorial - Discharge Air Flow in the *Rocky Tutorial Guide*

Tip:

You can Use Colored Directional Handles (p. 126) to change the shape and location.

See the images and table below to learn more about the Plane User Process.

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

Figure 5.138: Plane User Process applied to the Particles entity

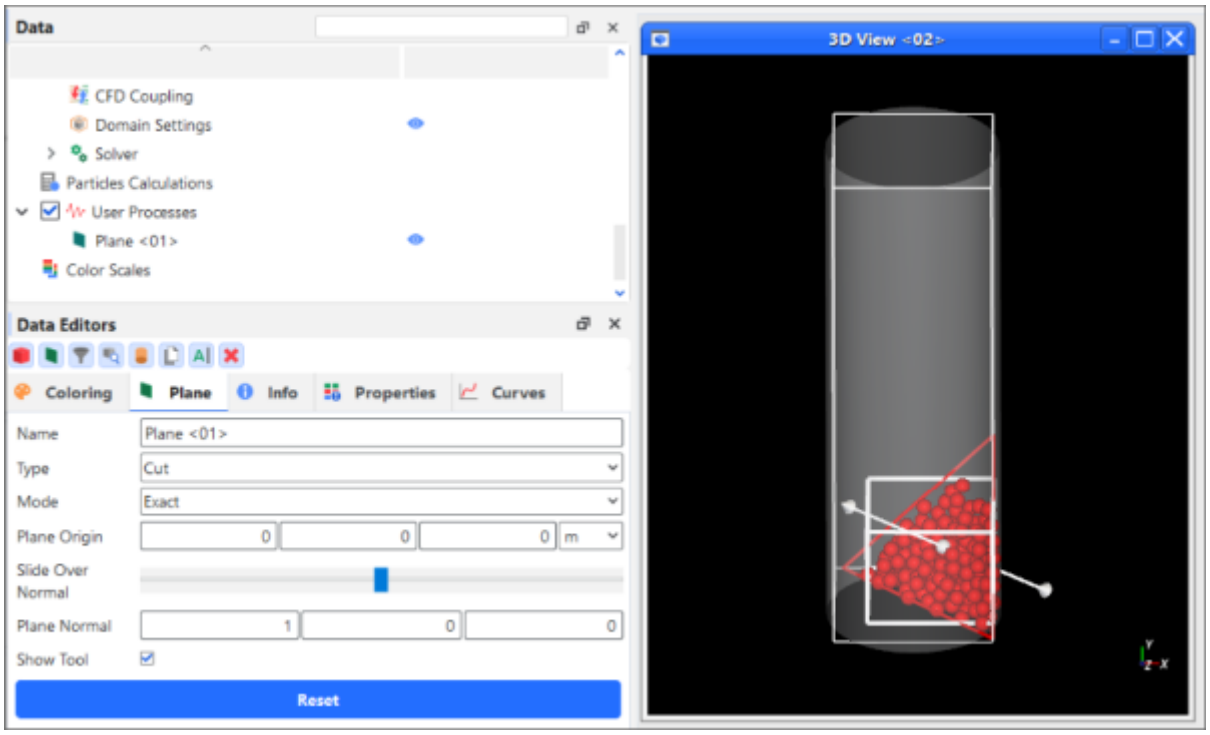


Figure 5.139: Plane User Process applied to a geometry component

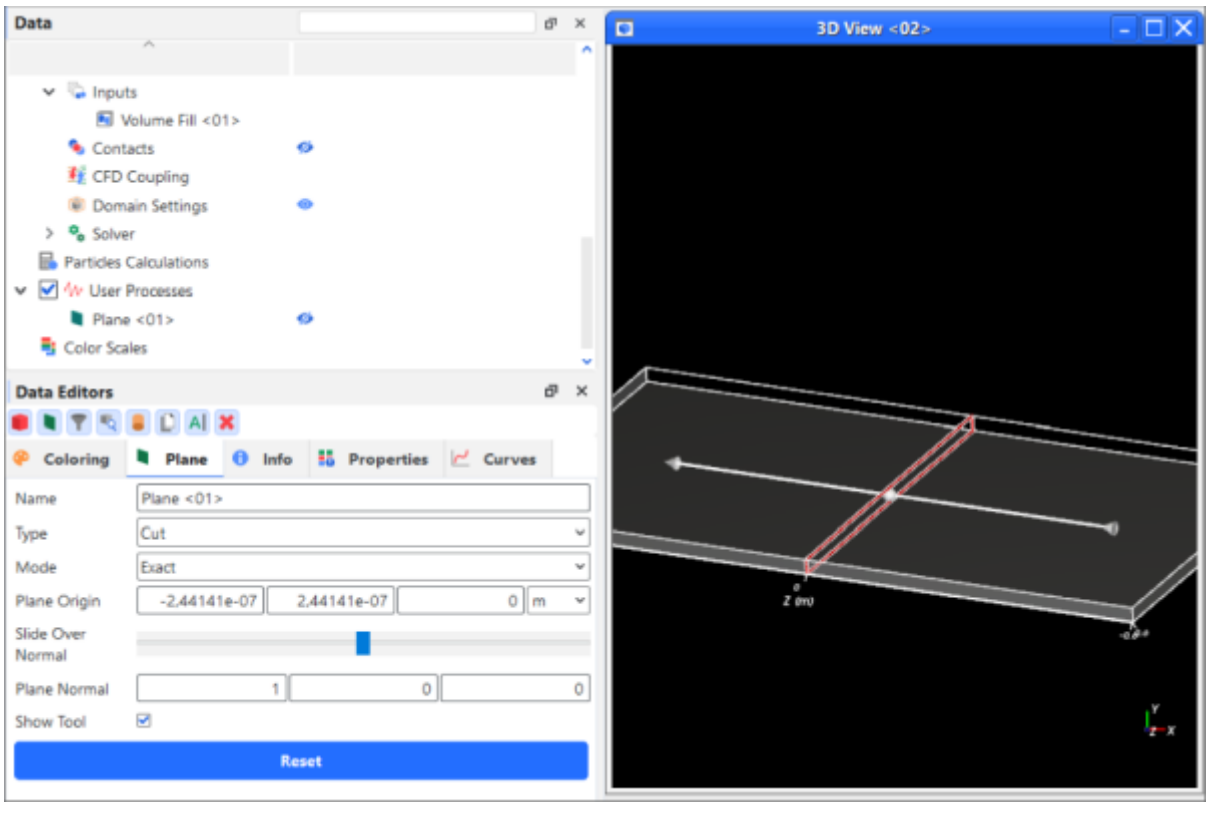


Table 1: Plane User Process parameter options

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

Setting	Description	Range
Name	Enables you to specify a unique identifier for the User Process. **Note:** This same name will appear in the **Data** panel under **User Processes**.	No limit
Type	For Planes based on individual geometries or 1-Way LBM, this determines how the Plane is used to define the analysis space. Specifically: • **Cut**: Includes results immediately surrounding the Plane. • **Clip**: Includes all results behind the plane. **Note:** For all other entities, the Plane Type is always Clip.	Cut; Clip
Mode	For Planes based on individual geometries or 1-Way LBM, this determines how much of the affected geometry *triangles* or LBM Air Flow cells are included in the results. Specifically: • **Exact**: Includes only the exact (partial) portion of any triangles or cells affected by the Plane. Note: For all other entities, the Plane Mode is always Exact. • **Select**: Includes the entire (whole) portion of any triangles or cells affected by the Plane.	Exact; Select
Plane Origin	The coordinate location of the Plane's origin as as specified in the X, Y, and Z directions.	Any value but display is limited by the boundaries of the entity
Slide Over Normal	Changes the location of the plane along the **Plane Normal** direction.	Values limited by slider
Plane Normal	The normal vector, often simply called the "normal" to a surface, is a vector perpendicular to the plane at a given point. The given X, Y, and Z vector components define the plane's outward normal. The domain considered for computation is the remaining part of the original domain after it is cut or clipped with the plane being defined.	No limit but values entered will be normalized

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

Setting	Description	Range
Show Tool	Enables the Plane, its Normal arrow, and directional handles to be displayed in the 3D View window.	Turns on or off

What would you like to do?

- Analyze Data with a Clip Plane (p. 1201)
- Analyze Data with a Cut Plane (p. 1202)
- Filter Views and Data with User Processes (p. 1174)
- Use Colored Directional Handles (p. 126)

Analyze Data with a Clip Plane

1. From the Data panel, right-click the entity (p. 1395) for which you want to analyze data, point to **Processes**, and then choose **Plane**. A new Plane entry appears under **User Processes** in the **Data** panel.
2. From the **Data Editors** panel, on the **Plane** tab, do one of the following: - For Planes based upon geometry components or 1-Way LBM, do all of the following:
   a. From the **Type** list, select **Clip**.
   b. From the **Mode** list, choose the level of detail you want.
   c. Enter the rest of the options you want.
   - For Planes based upon Particles or 1-Way Fluent, enter the options you want. Note: For these kinds of entities, Clip is automatically the Plane Type and Exact is the Mode.

Tip:

In the 3D View window (see also Create and Modify a 3D View (p. 1040)), you can also move, rotate, and resize the Plane by clicking and dragging the colored directional handles. (See also Use Colored Directional Handles (p. 126).)

See Also:

- About the Plane User Process (p. 1195)
- Analyze Data with a Cut Plane (p. 1202)
- Use Colored Directional Handles (p. 126)

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

Analyze Data with a Cut Plane

Note:

For Planes based upon Particles or 1-Way Fluent, a Clip Plane of Exact Mode is automatically set. (See also Analyze Data with a Clip Plane (p. 1201).)

1. From the Data panel, right-click the individual geometry component, 1-Way LBM, or User Process based on one of these entities (p. 1395) for which you want to analyze data, point to **Processes**, and then choose **Plane**. A new Plane entry appears under **User Processes** in the **Data** panel.
2. From the **Data Editors** panel, on the **Plane** tab, do all of the following:
   a. From the **Type** list, select **Cut**.
   b. From the **Mode** list, choose the level of detail you want.
   c. Enter the rest of the options you want.

Tip:

In the 3D View window (see also Create and Modify a 3D View (p. 1040)), you can also move, rotate, and resize the Plane by clicking and dragging the colored directional handles.

See Also:

- About the Plane User Process (p. 1195)

5.5.6. Analyze Data by Property Type

When you analyze data by property type, you reduce the data to only the particular calculation in which you are interested. This is accomplished when you apply the Filter User Process to a simulation entity (p. 1395) and then choose which property that you want to analyze. The properties you choose are identical to the ones listed on the Properties tab (see also About Properties (p. 933)).

What would you like to do?

- Learn more About the Filter User Process (p. 1202)
- Analyze Data with the Filter User Process (p. 1208)

About the Filter User Process

The Filter User Process enables you to limit your results by a certain property—the same ones listed on the Properties tab. In this way, you can focus your selection to only the calculation and data ranges you want. This method is useful for visualizing only the highest wear points on a conveyor, for example, or for seeing the mill location of the particles that have broken the least. It is also quite useful for separating particles into groups of only a particular property range, such as particle size, for more focused analyses.

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

This kind of User Process can be applied to individual geometry components, Particles, 1-Way LBM, 1-Way Fluent, or any other User Process created from these entities (p. 1395). However, because this kind of User Process is based upon calculations, you can only analyze data for the time steps that have been calculated.

The **Mode** setting of the Filter User Process affects how much of the component being analyzed (*triangle*, *cell*, *block*) will be displayed. **Select** displays the whole component containing the selected property (Figure 5.141: Filter User Process with Select Mode applied to a Eulerian Statistics User Process - Particles and Eulerian Statistics (On/Off) (p. 1204)); **Cut** displays only the portion of the component that contains the selected property (Figure 5.143: Filter User Process with Cut Mode applied to a Eulerian Statistics User Process - Particles and Eulerian Statistics (On/Off) (p. 1205)), which can provide a more realistic display of the particle flow.

Figure 5.140: Filter User Process with Select Mode applied to a Eulerian Statistics User Process - Data Editors Panel

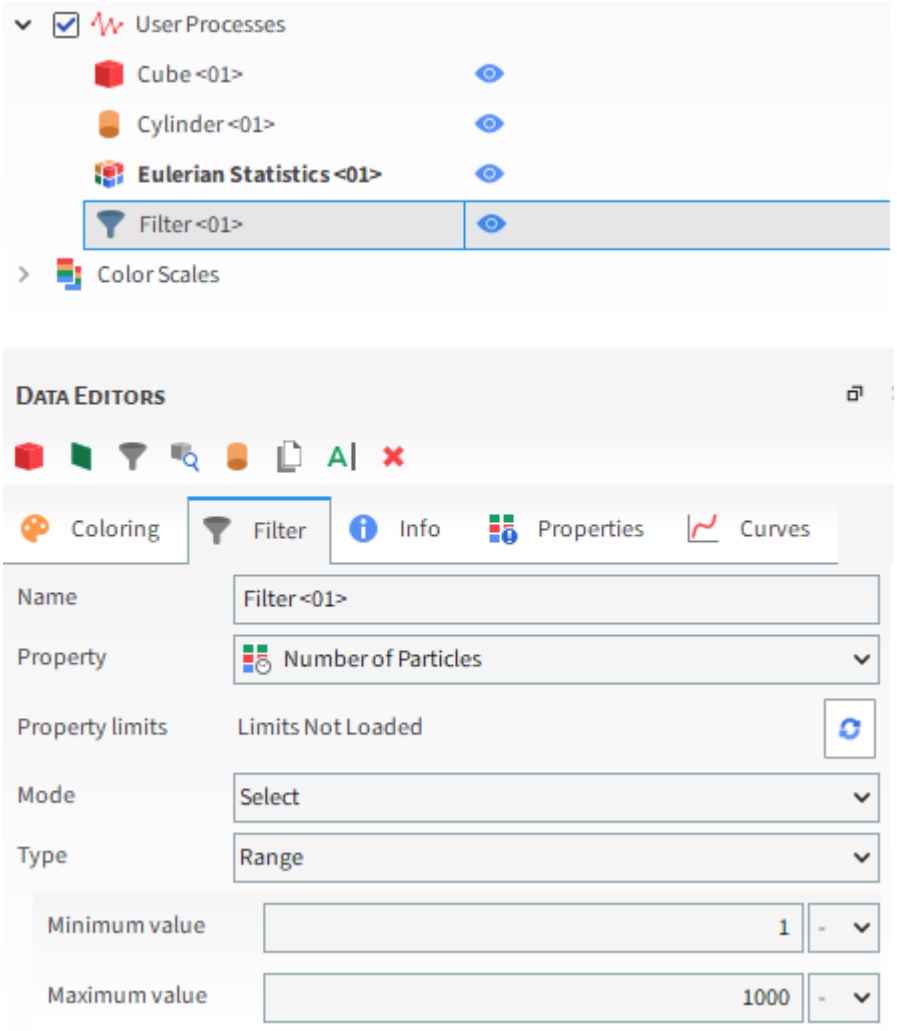


Figure 5.141: Filter User Process with Select Mode applied to a Eulerian Statistics User Process - Particles and Eulerian Statistics (On/Off)

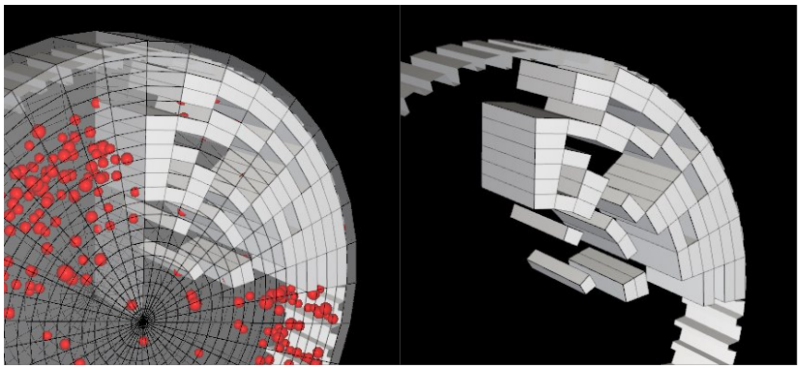
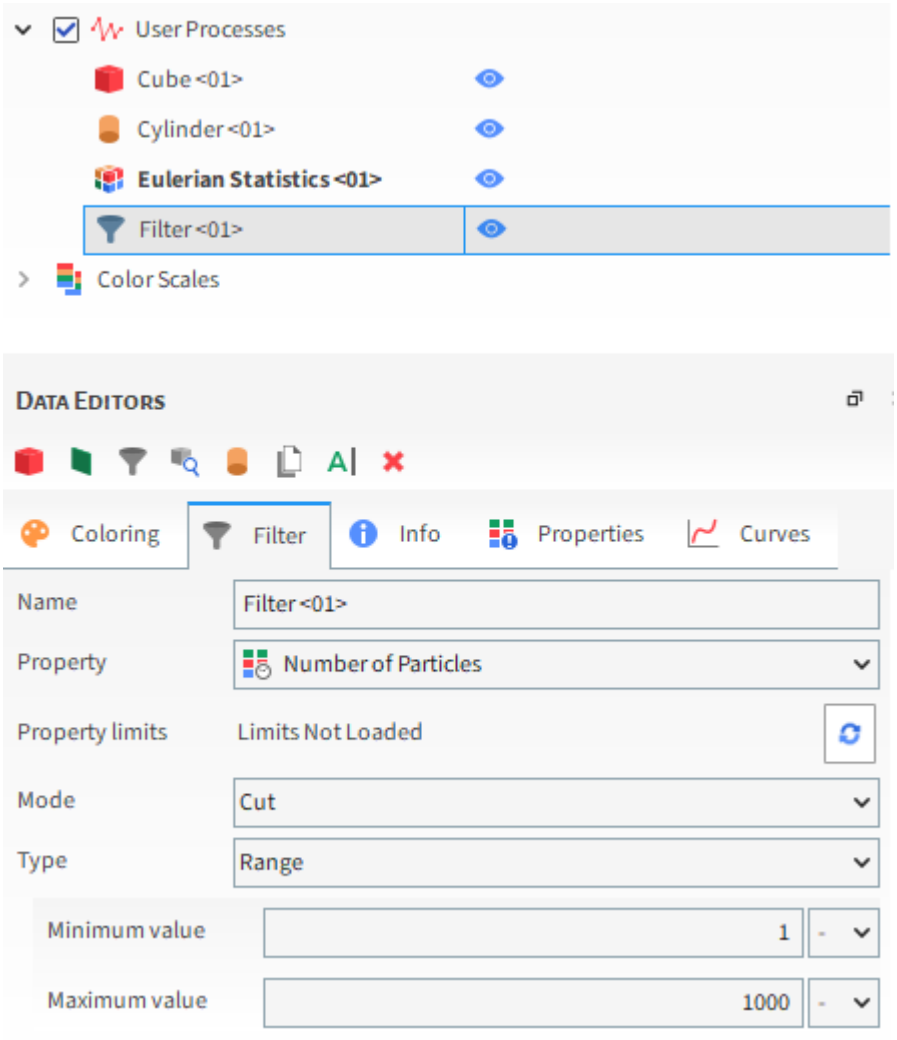
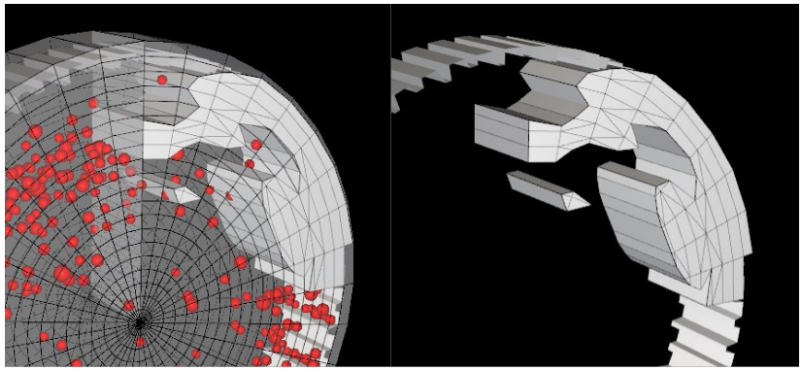


Figure 5.142: Filter User Process with Cut Mode applied to a Eulerian Statistics User Process - Data Editors Panel



Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

Figure 5.143: Filter User Process with Cut Mode applied to a Eulerian Statistics User Process - Particles and Eulerian Statistics (On/Off)



Tip:

To see walk-through examples of Filter User Processes being used to analyze simulation results, refer to the following Tutorials:

- Tutorial - Vibrating Screen in the *Rocky Tutorial Guide*
- Tutorial - Drop Weight Test in the *Rocky Tutorial Guide*
- Tutorial - High Pressure Grinding Roll (HPGR) in the *Rocky Tutorial Guide*

FILTER USER PROCESS PARAMETER DEFINITIONS

See the image and table below to learn more about the Filter User Process.

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

Figure 5.144: Filter User Process applied to a custom conveyor - Data Editors Panel

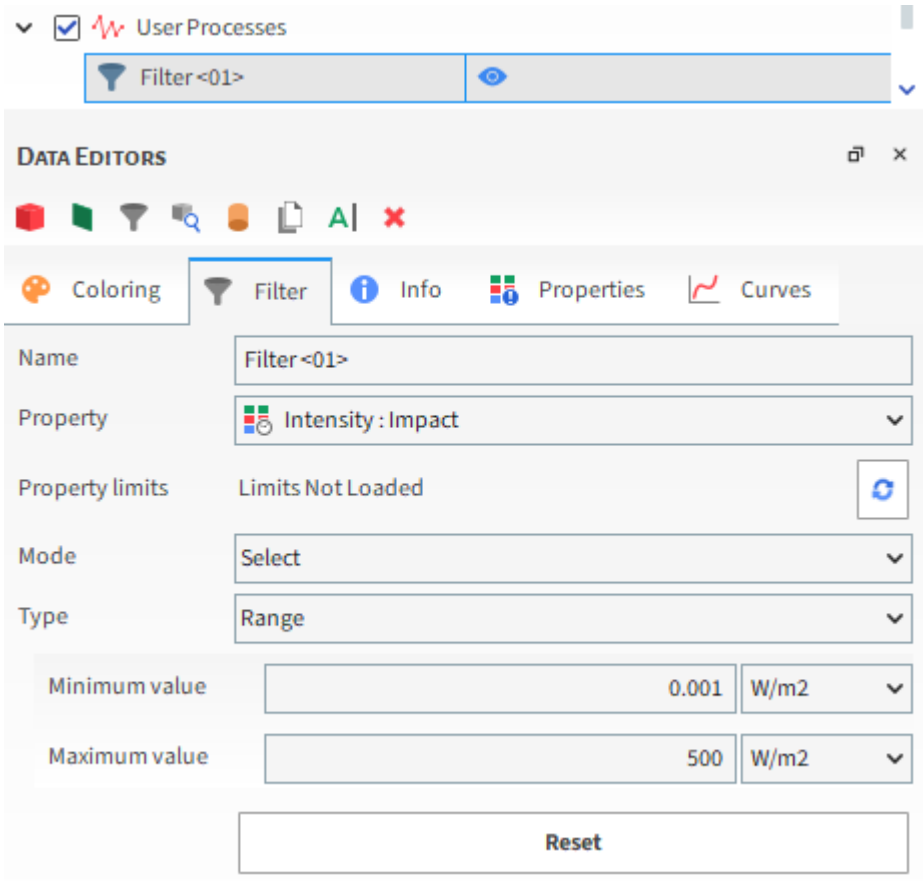


Figure 5.145: Filter User Process applied to a custom conveyor

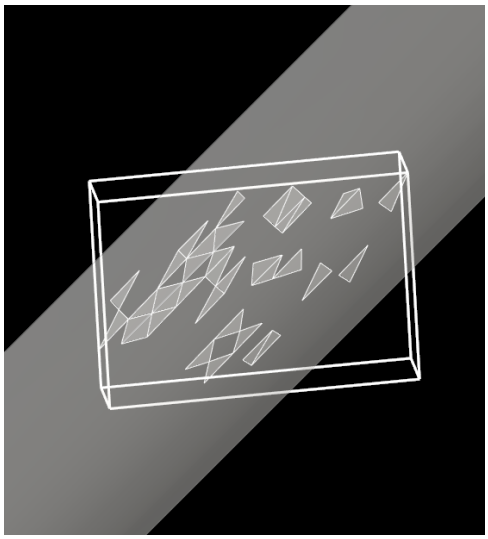


Table 1: Filter User Process parameter options

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

Setting	Description	Range
Name	Enables you to specify a unique identifier for the User Process. **Note:** This same name will appear in the **Data** panel under **User Processes**.	No limit
Property	Lists the available properties that you can apply. This is the same list as displayed on the entity's Properties tab. (See also About Properties (p. 933)).	Automatically generated
Property limits	Displays the minimum and maximum values calculated for the selected **Property** based upon the available time steps. This range is useful when entering the Value or Range. **Tip:** If the limits are not yet loaded, click the **Load Limits** button to have Rocky perform the necessary calculations.	Display only
Mode	For Filter User Processes based on individual geometries, 1-Way LBM, or Eulerian Statistics, this determines how much of the affected geometry *triangle* (p. 1395), LBM Air Flow *cell* (p. 1389), or Eulerian *triangle* (p. 1395) are included in the results shown. Specifically: • **Cut**: Includes only the triangle vertices, cells, or block volume portions affected by the Property. • **Select**: Includes the entire (whole) portion of any triangles, cells, or blocks affected by the Property.	Cut; Select
Type	Determines how the values for the property are limited. Specifically: • **Value**: Includes only the results for the exact value you specify in the **Cut Value** box. • **Range**: Includes all results between the **Minimum value** and **Maximum value** values you specify.	Value; Range
Cut Value	When the **Type** of **Value** is selected, this is the single, exact value by which you want the property results limited.	Any value

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

Setting	Description	Range
Minimum value	When the **Type** of **Range** is selected, this is the lowest value by which you want the property results limited. **Tip:** Use the **Property limits** to help you determine what range values to set.	Any value
Maximum value	When the **Type** of **Range** is selected, this is the highest value by which you want the property results limited. **Tip:** Use the **Property limits** to help you determine what range values to set.	Any value

See Also:

- Analyze Data with the Filter User Process (p. 1208)
- Filter Views and Data with User Processes (p. 1174)

Analyze Data with the Filter User Process

1. From the **Data** panel, right-click the entity (p. 1395) for which you want to analyze data, point to **Processes**, and then choose **Property**. A new Property entry appears under **User Processes** in the **Data** panel.
2. From the **Data Editors** panel, on the **Property** tab, enter the options you want.

Tip:

To revert values back to the default settings, which uses the **Property limits** to calculate middle-range values, click the **Reset** button.

See Also:

- About the Filter User Process (p. 1202)
- About Properties (p. 933)

5.5.7. Analyze the Particles Trajectory

When you analyze the particles trajectory, you are having Rocky translate the path of the already-calculated particles into a collection of 3D curves. This is accomplished by applying the Particles Trajectory User Process to the Particles entity.

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

What would you like to do?

- Learn more About the Particles Trajectory User Process (p. 1209)
- Analyze Data with the Particles Trajectory User Process (p. 1210)

About the Particles Trajectory User Process

The Particles Trajectory User Process enables you to more easily see the flow of your particles over time by translating their paths into 3D curves. This kind of User Process can be applied only to the Particles entity (p. 1395) or other User Processes based upon the Particles entity.

Tip:

To see walk-through examples of the Particle Trajectory User Process, review the following Tutorials:

- Tutorial - SAG Mill in the *Rocky Tutorial Guide*
- Tutorial - **Bucket Conveyor** in the *Rocky Tutorial Guide*

Use the image and table below to help you understand how to use the Particles Trajectory User Process.

Figure 5.146: Particle flow (top) compared with Particles Trajectory User Process (bottom).

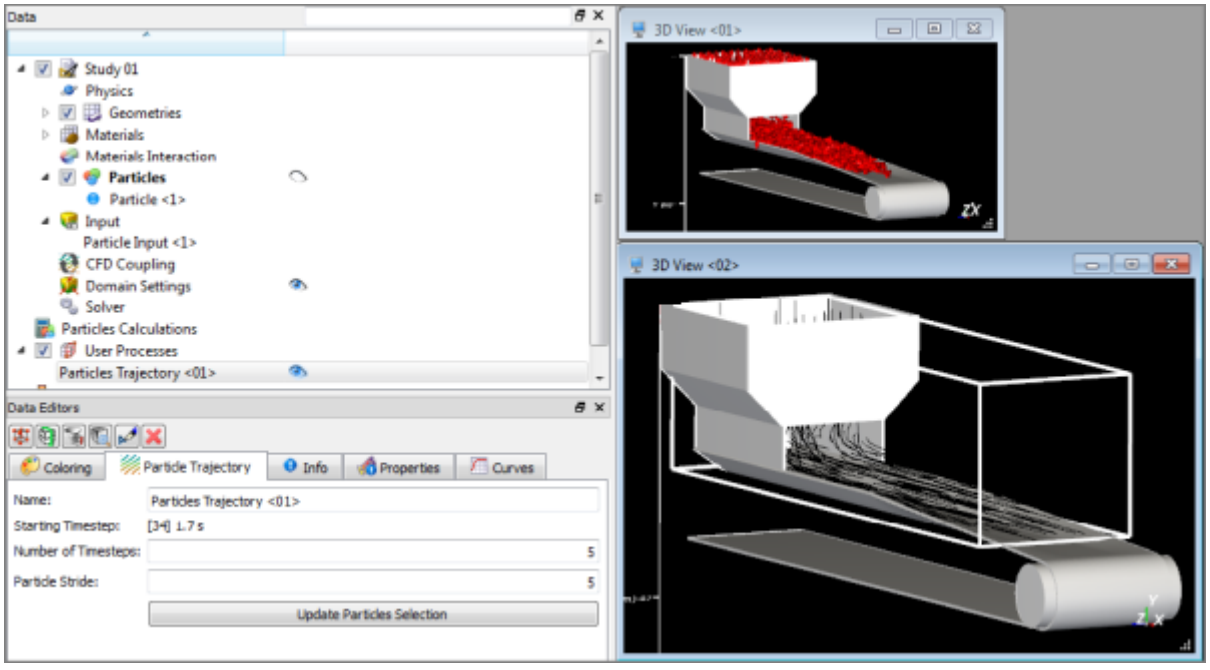


Table 1: Particles Trajectory User Process parameter options

Setting	Description	Range
Name	Enables you to specify a unique identifier for the User Process. **Note:** This same name will appear in the **Data** panel under **User Processes**.	No limit
Starting Time	Displays the current time that you've set in the Time toolbar. (See also About the Time Toolbar (p. 1279).)	Display only
Number of Intervals	The amount of additional intervals beyond the **starting time step** to include in calculations. A lower value will calculate faster but will be less detailed. A higher value will be more detailed but will take more time to process. **Tip:** Ensure that your **Starting Time** is at least the **Number of Intervals** lower than the final time calculated for your simulation.	Positive values
Particle Stride	One out of this number of particles will be picked when calculating trajectories. The lower the number, the more particles will be included in the calculation. Example: Choosing a **Particle Stride** of 5 means that 1 particle out of every 5 will be used in the trajectory calculation.	Positive values

See Also:

- Analyze Data with the Particles Trajectory User Process (p. 1210)

Analyze Data with the Particles Trajectory User Process

1. Ensure you have a 3D View window selected showing the area you want to analyze. (See also Create and Modify a 3D View (p. 1040).)

2. From the Time toolbar (see also About the Time Toolbar (p. 1279)), select the time step from which you want Particle Trajectory calculations to start. Tip:Ensure that you choose a time step from the toolbar that is at least the **number of time steps** (see also Analyze Data with the Particles Trajectory User Process (p. 1210)) lower than the final time step you have available for your simulation. Note: The value you choose here will be your **starting time step** value.

3. From the **Data** panel, right-click the **Particles** entity (p. 1395) for which you want to analyze data, point to **Processes**, and then choose **Particles Trajectory**. A new Particles Trajectory entry appears under **User Processes** in the **Data** panel.

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

4. From the **Data Editors** panel, on the **Particle Trajectory** tab, enter the values you want, and then click **Update Particles Selection**. The 3D View window displays the particles trajectory.

Tip:

If you do not immediately see the trajectory vectors in the 3D View window, you might need to first hide the Particles data. (See also Show/Hide Components by Using Eye icons and Checkboxes (p. 113).)

See Also:

- Analyze Data with the Particles Trajectory User Process (p. 1210)

5.5.8. Inspect an Individual Cell

When you apply the Cell Inspector User Process to a simulation entity (p. 1395) (e.g., individual geometry component, Particles, 1-Way LBM, 1-Way Fluent) or an Eulerian Statistics User Process, you are able to select and inspect the details of an individual cell (p. 1389) (triangle, block, or node) making up that particular entity.

What would you like to do?

- Learn more About the Cell Inspector (p. 1211)
- Inspect an Individual Cell (Triangle, Block, or Node) for an Entity (p. 1218)
- Learn more About Eulerian Statistics (p. 1226)

See Also:

- Filter Views and Data with User Processes (p. 1174)

About the Cell Inspector

The Cell Inspector is a User Process you can apply to specific simulation entities (p. 1395) that enables you to select and inspect the details of an individual cell (p. 1389) making up that particular entity. Cell Inspectors can be applied to an individual geometry component, Particles, 1-Way LBM, or 1-Way Fluent; or to a User Process shape (Cube, Cylinder, or Polyhedron) or Eulerian Statistics User Process created from the Particles entity.

For geometry components, each cell inspected is made up of an individual triangle (p. 1395); for Air Flow, this is an individual cell (p. 1389) ; for 1-Way Fluent and Particles, this is an individual node (p. 1393); and for Eulerian Statistics, this is an individual block (p. 1389).

Only one cell may be inspected at any one time per Cell Inspector User Process, but you can change the location of the cell being inspected by using the **ID** field in the **Data Editors** | **Inspector**, tab or by double-clicking a new cell from within any 3D View window. You may have as many separate Cell Inspector User Processes as you like.

As with all User Processes, any Cell Inspector User Process created is considered to be linked to the original entity and will display only the chosen subset of information. (See also See Input/Output Relationships in the Data Panel When Component Names are Bold (p. 114)).

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

Once you have selected the cell you want to inspect, you can then use the **Properties** tab to display individual values for the properties you are interested in, or can choose to create a Time Plot showing all output time values for that particular property.

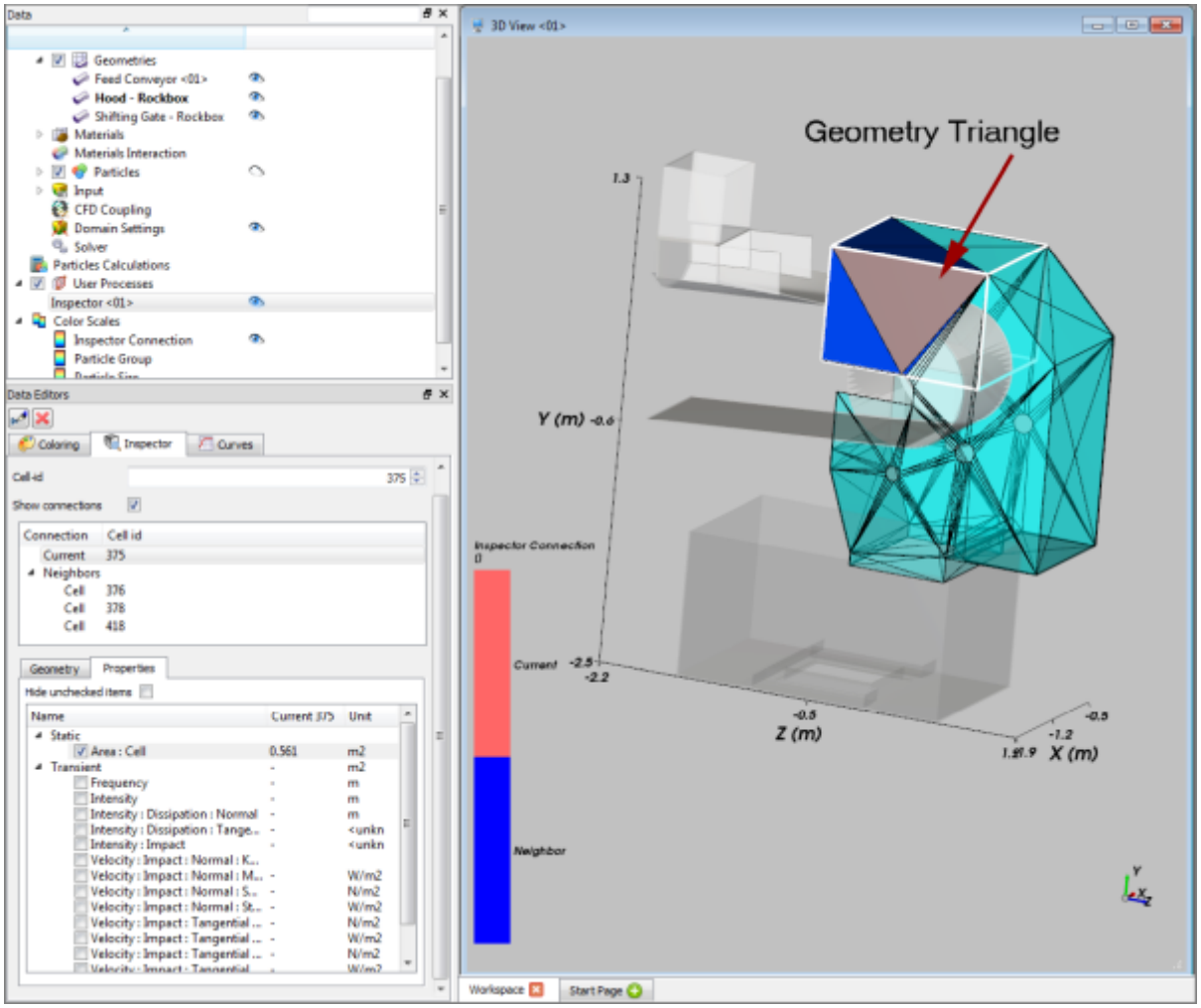
Tips:

- You may also choose to use the separate **Inspection** panel, which can be opened from the **Tools** menu (Figure 5.147: Inspector tab on the Data Editors panel for a Geometry Cell Inspector User Process (left); Geometry triangle selected for inspection (right) (p. 1212)). This panel provides access to all the same information available on the Inspection tab of the Data Editors panel, but enables you to switch more easily between your various Cell Inspector User Processes.
- To see a walk-through example of using a Cell Inspector User Process, review:

Tutorial - Conical Dryer in the *Rocky Tutorial Guide*.

Use the images and table below to understand more about applying and using the Cell Inspector.

Figure 5.147: Inspector tab on the Data Editors panel for a Geometry Cell Inspector User Process (left); Geometry triangle selected for inspection (right)



Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

Figure 5.148: Inspector tab on the Data Editors panel for a Particles Cell Inspector User Process (left); individual whole particle selected for inspection (right)

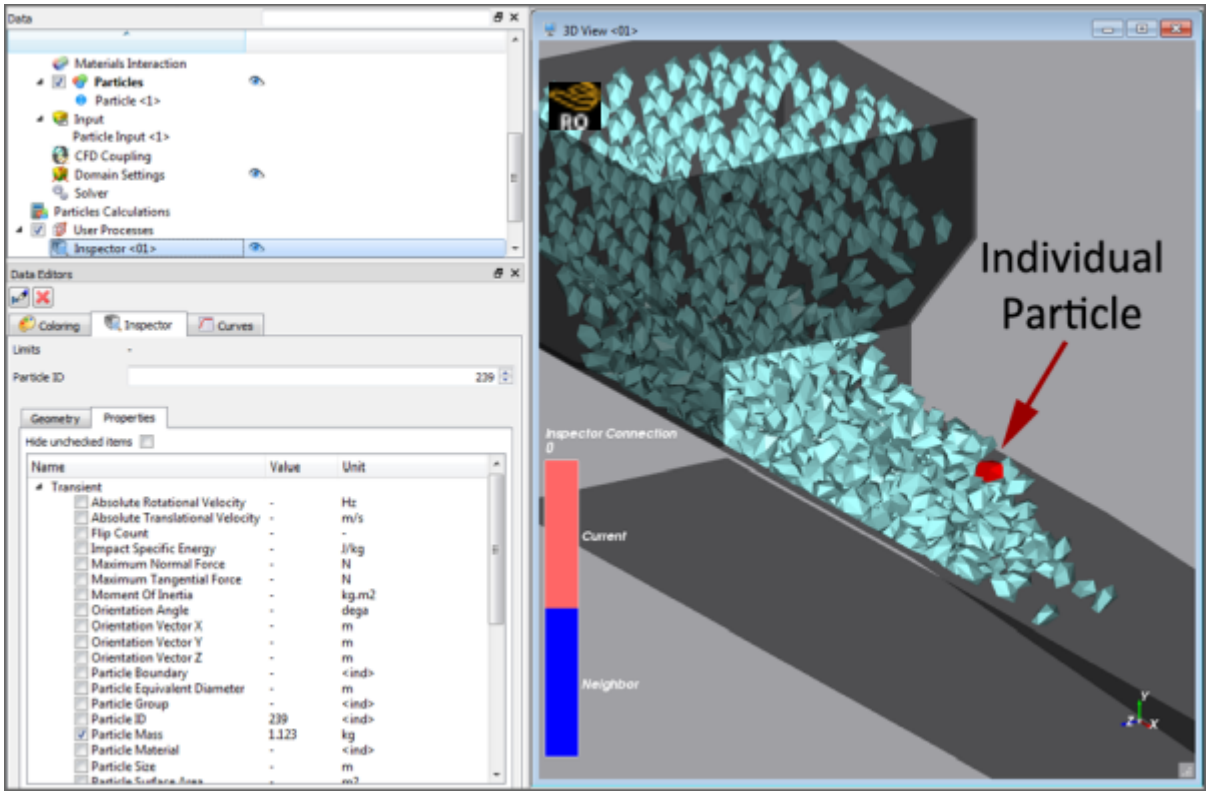


Figure 5.149: Inspector tab on the Data Editors panel for a Particles Cell Inspector User Process (left); individual particle fragment selected for inspection (right)

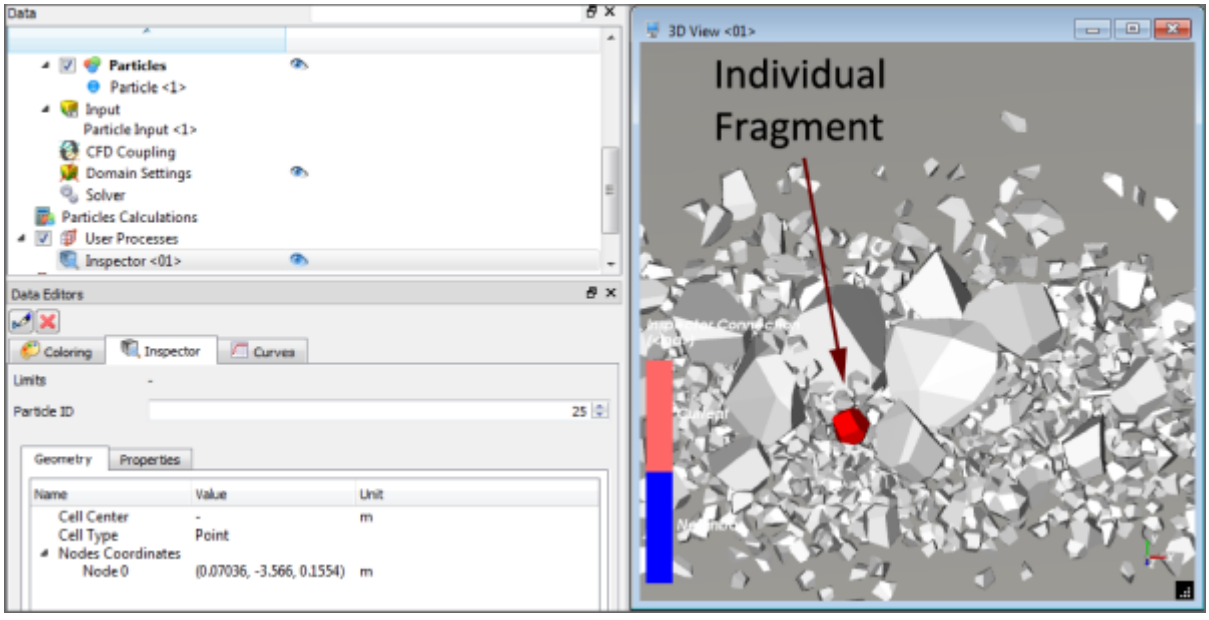
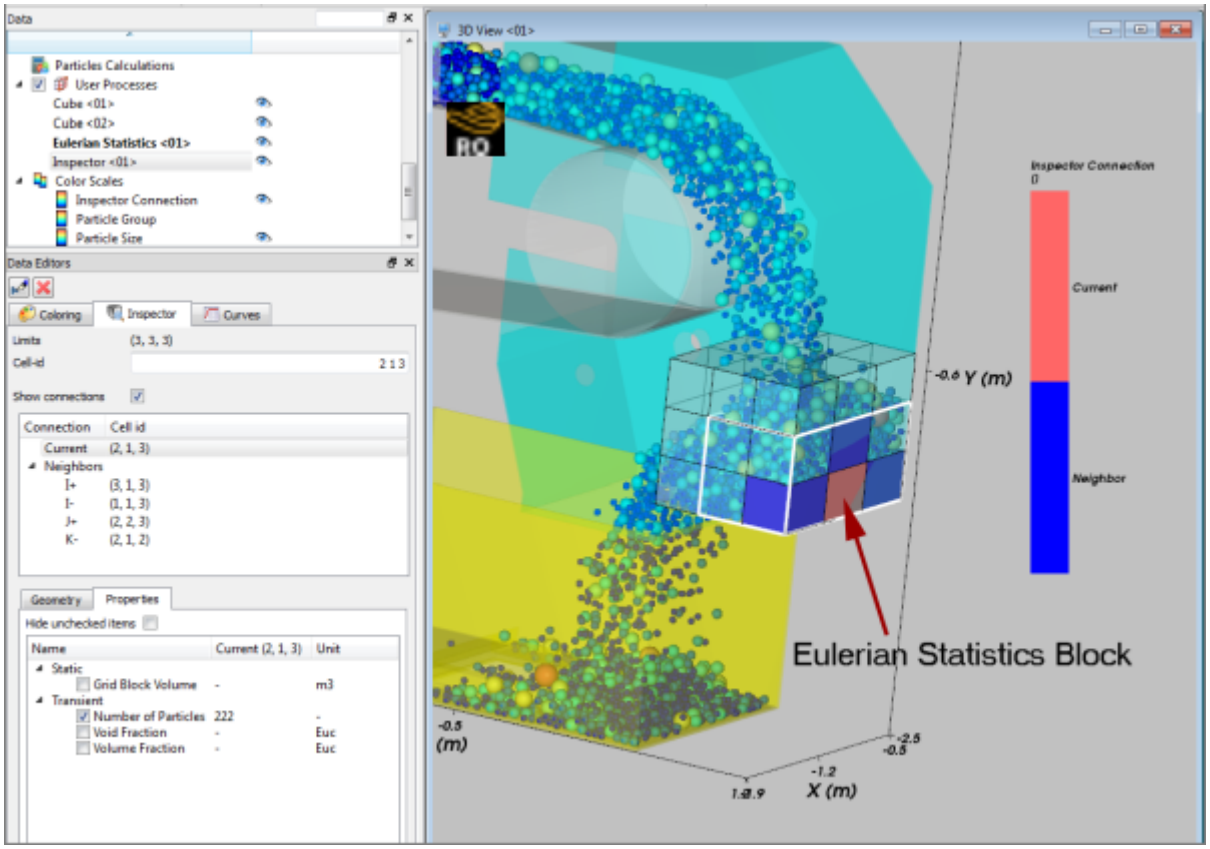
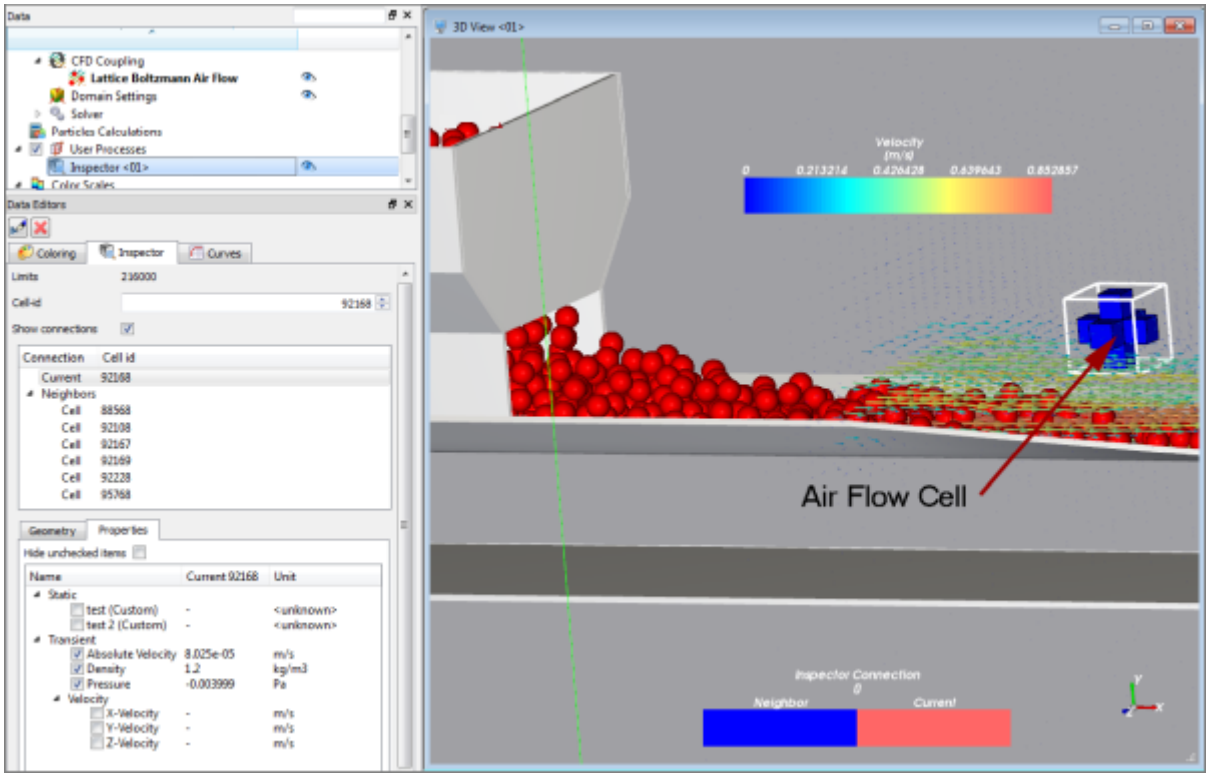


Figure 5.150: Inspector tab on the Data Editors panel for a Eulerian Statistics Cell Inspector User Process (left); Eulerian Statistics block selected for inspection (right)



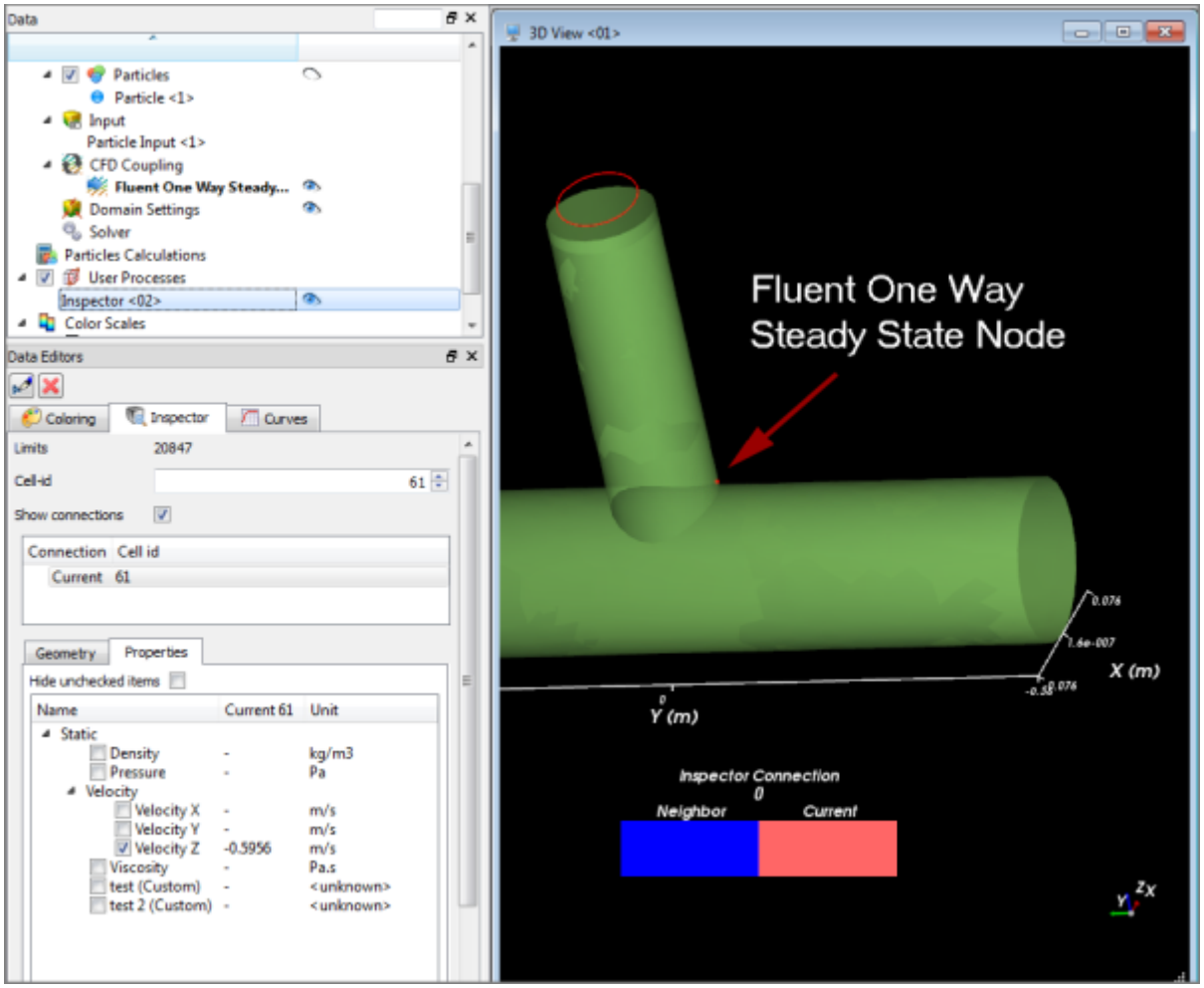
Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

Figure 5.151: Inspector tab on the Data Editors panel for a 1-Way LBM Air Flow Cell Inspector User Process (left); Air Flow cell selected for inspection (right)



Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

Figure 5.152: Inspector tab on the Data Editors panel for a 1-Way Fluent Inspector User Process (left); 1-Way Fluent node selected for inspection (right)



Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

Figure 5.153: Inspector panel showing a Geometry Cell Inspector User Process

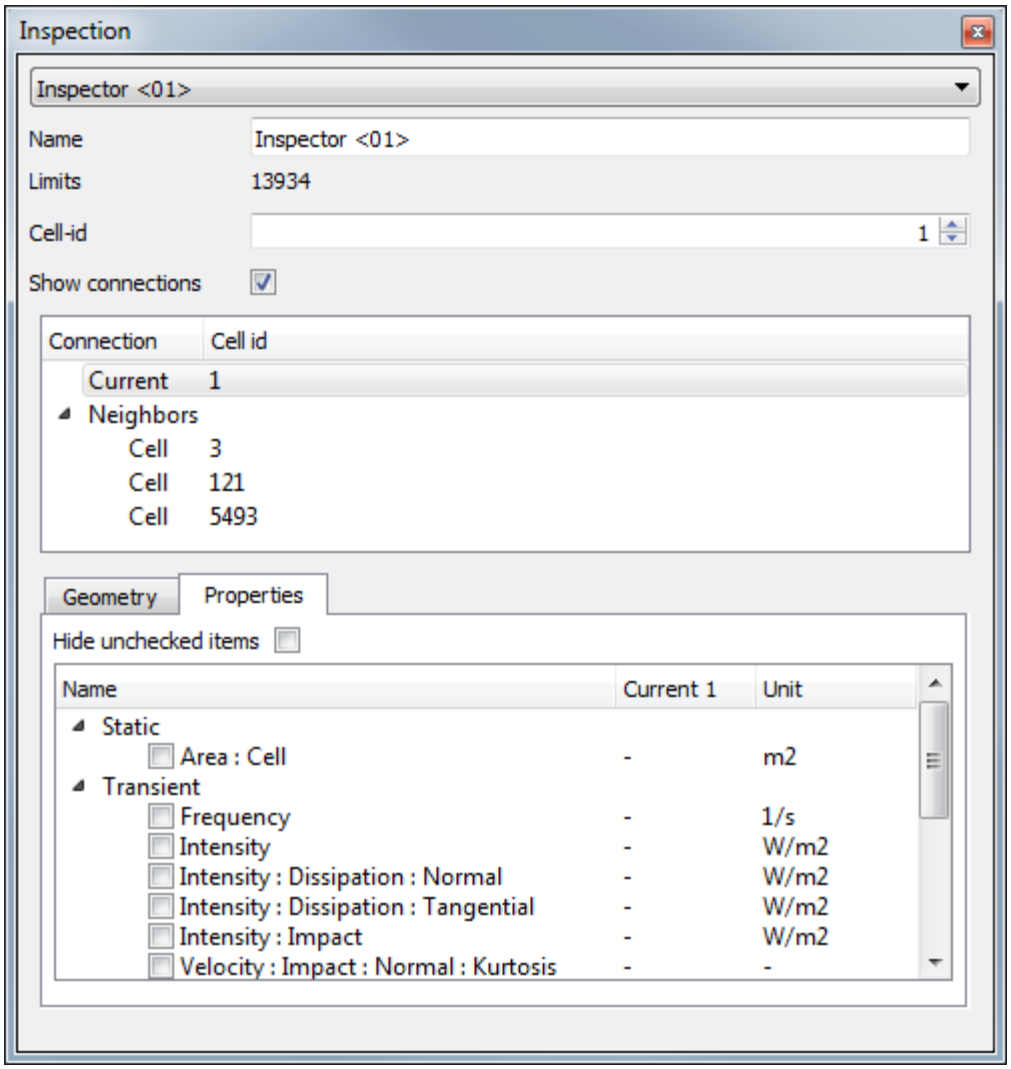


Table 1: Inspector options

Setting	Description	Range
Name	Enables you to specify a unique identifier for the User Process. **Note:** This same name will appear in the **Data** panel under **User Processes**.	No limit
Limits	For all entities but Geometries and Particles, displays the highest ID currently available. This helps you limit the value you enter in the **ID** box.	Automatically generated
ID (Cell or Particle)	Displays the unique identifier (ID) associated with the selected cell. Specifically: • For Geometries, this is the triangle ID.	Inputs restricted by **Limits**

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

Setting	Description	Range
	• For Particles, this is the particle ID. • For Eulerian Statistics, this corresponds to the number of blocks in the three local x, y, z directions (or i, j, k on the Cartesian coordinate system) respectively. (See also About Eulerian Statistics (p. 1226)). • For 1-Way LBM, this is the Air Flow cell ID. (See also About Using the 1-Way LBM Method (p. 771)). • For 1-Way Fluent, this is the fluid node.	
Show Connections	For all entities but Particles and 1-Way Fluent, this enables the IDs of the cells surrounding the selected cell (its Neighbors) to be displayed. **Tip:** Double-clicking a Neighbor ID will make that cell the selected cell. Repeatedly selecting Neighbor cells in the same direction will move the cell selection in that direction.	Turns on or off

What would you like to do?

- Inspect an Individual Cell (Triangle, Block, or Node) for an Entity (p. 1218)

See Also:

- Filter Views and Data with User Processes (p. 1174)
- Use Properties and Curves (p. 931)
- Graph (Plot or Histogram) a Data Set Within Rocky (p. 1101)

Inspect an Individual Cell (Triangle, Block, or Node) for an Entity

1. From the **Data** panel, right-click the simulation entity (p. 1395) for which you want to analyze an individual cell (p. 1389) (triangle, block, node, or air flow cell (p. 1389)).

2. From the right-click menu, point to **Processes**, and then click **Cell Inspector**. An **Inspector** entry appears under **User Processes**.

1. From the **Data Editors** panel, do any or all of the following:

   - Use the **ID** field to change the position of the cell selected. Tip:This selection will also be visible in a 3D View window.

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

- If available, use the **Show Connections** check box to select whether details about the neighbor cells are visible or hidden.

Tip:

If you have **Show connections** enabled and there are cell IDs listed under **Neighbors**, you may double-click any Neighbor Cell to make that cell the selection.

---

2. After you have selected the cell you want, you can view data for that cell by doing any or all of the following:

   - From the **Properties** tab on the bottom half of the **Inspector** tab, select the properties that you want to view. The values for the selected time step and property are displayed on the tab.

   - Right-click any **Property** property, point to **Time Plot**, and then select one of the options displayed. Values for all available time steps for that property are displayed in the Time Plot.

Tips:

- For Geometries and Eulerian Statistics, you may also double-click the triangle (p. 1395) or block (p. 1389) you want to inspect from any 3D View window and a new Inspector User Process will be automatically created for the entity you chose. If there is already an Inspector User Process for that entity, Rocky will help you choose to update the cell location of the existing User Process, or create a new one. Note: Double-clicking a cell in a 3D View window does not currently work for Particles, 1-Way LBM, or 1-Way Fluent.

- When using the 3D View window, it helps to have **Edges** for the component turned on so that the cells can be more easily seen and selected. (See Also Create and Modify a 3D View (p. 1040).)

- When viewing Particle or 1-Way Fluent nodes in a 3D View window, it helps to hide Particles in the view. (See also Show/Hide Components by Using Eye icons and Checkboxes (p. 113).)

- After creating your Cell Inspector User Process, you may choose to use the Inspection panel instead of the Data Editors panel to change your settings, or to switch between Cell Inspector User Processes. To open the panel, from the **Tools** menu, click **Inspection**.

See Also:

- About the Cell Inspector (p. 1211)

- Use Properties and Curves (p. 931)

- Graph (Plot or Histogram) a Data Set Within Rocky (p. 1101)

5.5.9. Duplicate a User Process

When you duplicate a User Process, you make an exact copy that you can then use as a starting place for making a similar but slightly different analysis. Duplicating an already-defined User Process can be quicker than starting from a brand new User Process.

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

To duplicate a User Process:

1. Ensure that the User Process you want to duplicate is already created and set up the way you want.

2. From the **Data** panel, right-click the User Process you want to copy, point to **Processes**, and then choose **Duplicate**. An copy of the User Process appears under **User Processes**, the name of which is appended with a new number.

See Also:

- Filter Views and Data with User Processes (p. 1174)

5.5.10. Export User Process Geometry Triangles to an STL File

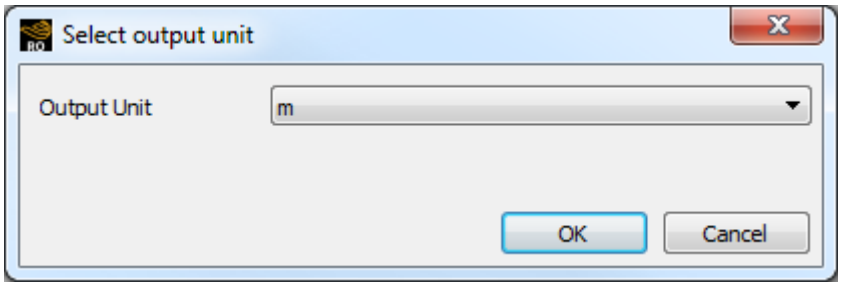
Being able to export a rendered geometry (p. 1394) component to an .stl file for use outside of Rocky extends to any (whole) individual triangles that are selected through a User Process that was created from a geometry component.

Important:

Because the export works only with whole geometry triangles, any triangles only partially included in a User Process selection will not be included in the export. This means that if your User Process selects only a portion of a geometry triangle, the resulting .stl file might be empty.

As with its parent geometry component, exporting the shapes of User Process geometry triangles enables you to select which units you want to use when exporting (as shown below).

Figure 5.154: Select Output Unit dialog



To export User Process geometry triangles to an STL file:

1. Ensure that you have set up the User Process based upon the geometry component to select the (whole) geometry triangles that you want to export.

2. From the **Data** panel, under **User Processes** right-click the process you want, point to **Export**, and then click **Rendered Geometry**.

3. From the **Select output unit** dialog, select from the **Output Unit** list the units you want, and then click **OK**.

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

4. From the **Select target STL file** dialog, click the drive or folder of the location to which you want to save the file.

5. In the **File name** box, enter a name for the file, and then click **Save**.

See Also:

- Filter Views and Data with User Processes (p. 1174)
- About Exporting Data and Images Out of Rocky (p. 1155)
- Export a Geometry Component to an STL File (p. 520)

5.5.11. Analyze Particles within a Time Selection

When you analyze Particles within a Time Selection, you are choosing to limit the Particles data by a particular time range. In this way, you can select only the Particles affecting a certain area of your simulation during a certain time frame for further analysis.

What would you like to do?

- Learn more About the Particles Time Selection User Process (p. 1221)
- Analyze Data with the Particles Time Selection User Process (p. 1225)

See Also:

- Filter Views and Data with User Processes (p. 1174)

About the Particles Time Selection User Process

The Particles Time Selection User Process enables you to limit the Particles data by a particular time range. In this way, you can select only the Particles affecting a certain area of your simulation during a certain time frame or period.

Particles Time Selection User Processes can be applied directly to the Particles entity, to any shape User Process created from the Particles entity (Cube, Cylinder, or Polyhedron), or to any Property or Plane User Process created from the Particles entity.

Tip:

To see walk-through examples using the Particles Time Selection User Process, review the following Tutorials:

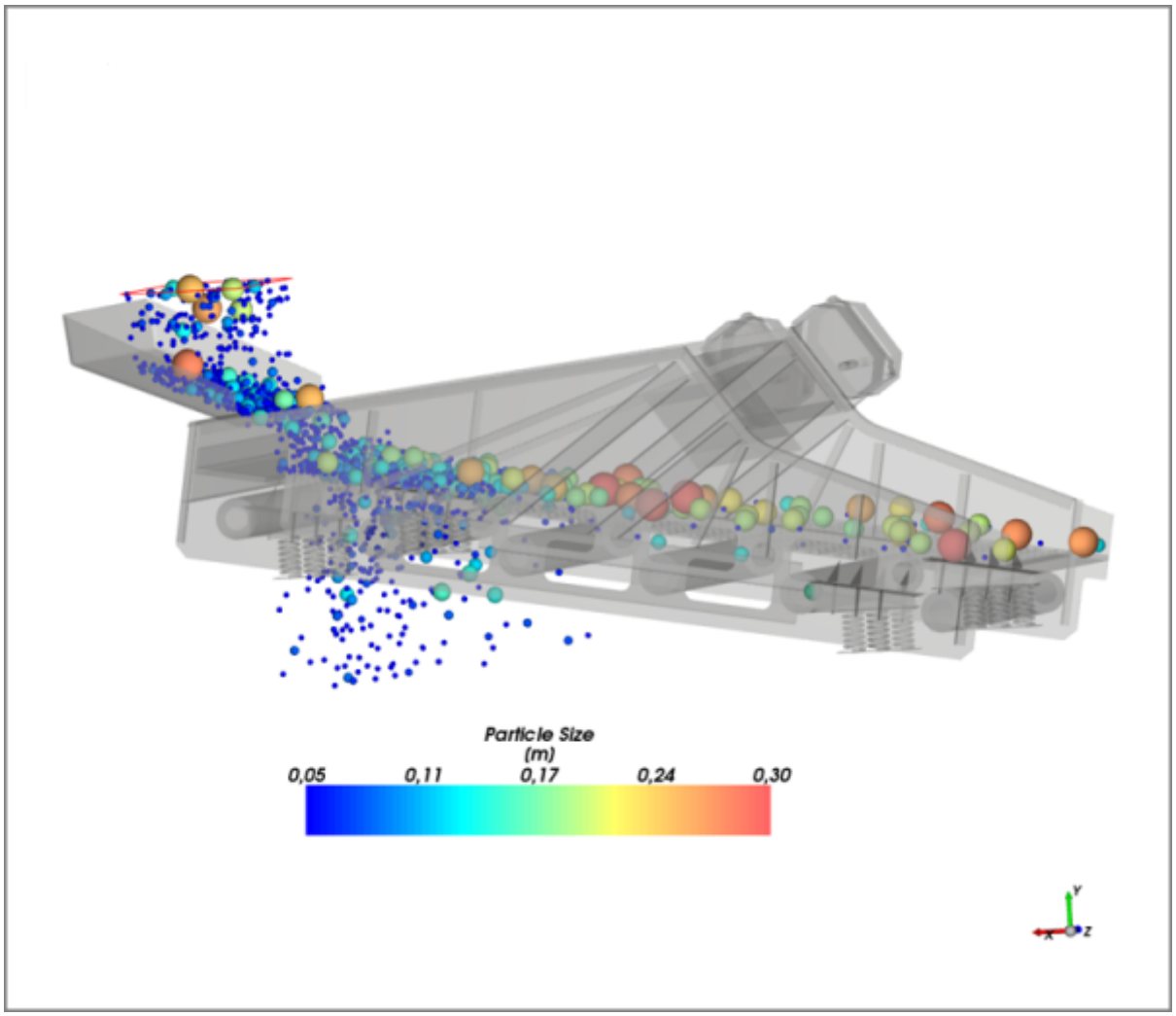
- Tutorial - Vibrating Screen in the *Rocky Tutorial Guide*
- Tutorial - High Pressure Grinding Roll (HPGR) in the *Rocky Tutorial Guide*
- Tutorial - Cone Crusher in the *Rocky Tutorial Guide*

TIME SELECTION EXAMPLE

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

For example, this method could be useful in cases where you want to analyze the sorting efficiency of a screening device designed to separate particles based upon their size (as shown below).

Figure 5.155: Example screening device designed to sort particles based upon their size

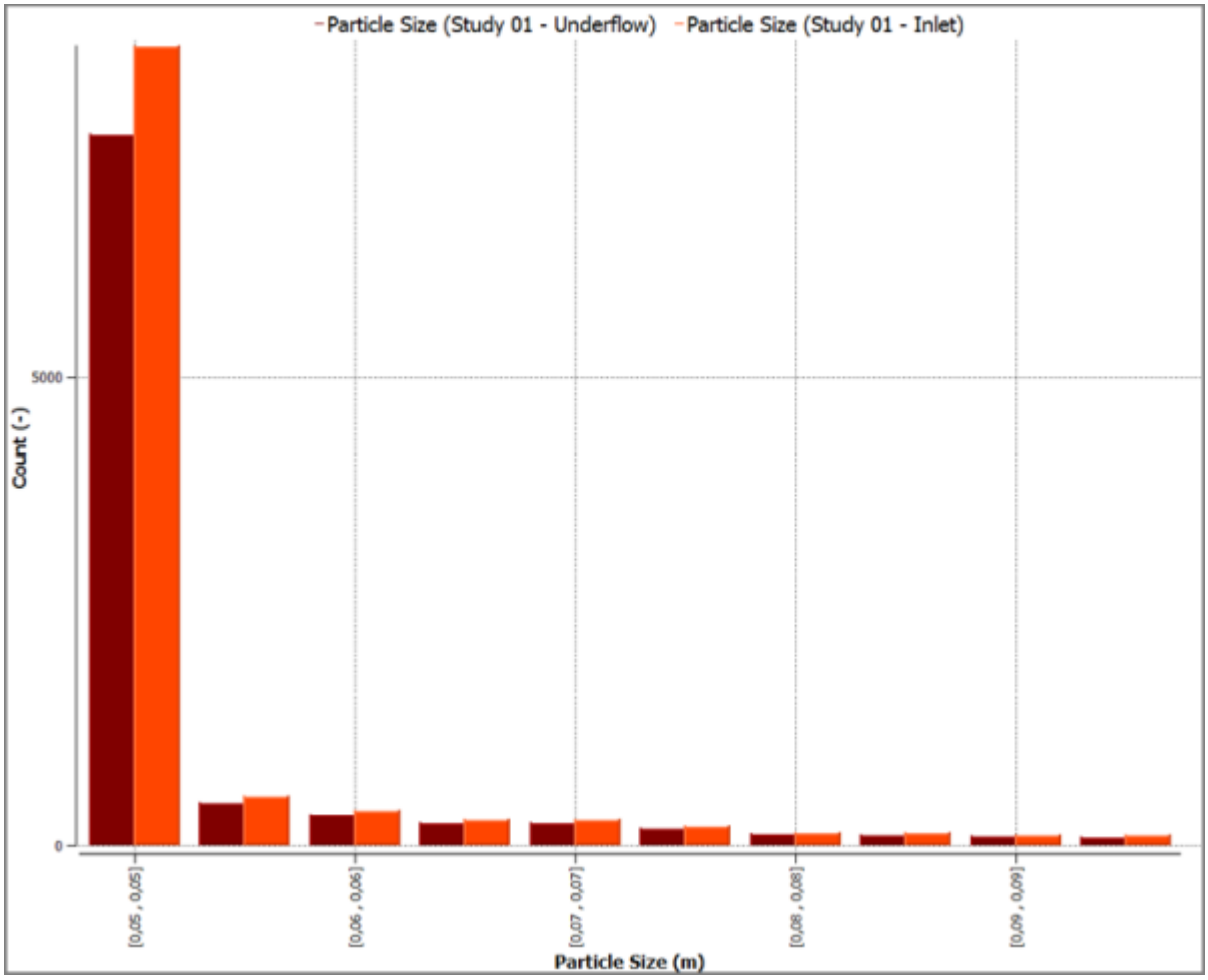


In this example, the smaller particles leave the screen from the bottom while the bigger particles that cannot pass through the smaller bottom holes roll until they reach the end of the device. By creating a Cube User Process (see also Analyze Data Within a Defined Shape (p. 1175)) below the bottom screen and then applying a Particles Time Selection User Process to the Cube, all of the particles that have entered the cube from time A to time B can be selected. Then by using Properties (see also About Properties (p. 933)), you can analyze the Particle Size of just this selection.

For example, you can create a histogram that compares the particle size distribution (PSD) of the particles that exited from the bottom compared to the PSD of the particles that have entered the domain (as shown below).

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

Figure 5.156: Example histogram showing the PSD of the particles that exited at the bottom (red) compared to the PSD of the particles that entered the domain (orange)



This kind of analysis shows that not all of the small particles exited from the bottom, perhaps indicating that the device has room for improvement in terms of its sorting efficiency.

TIME SELECTION AND BREAKAGE

If your project was processed with Rocky v4 or later and also had breakage modeling enabled (see also Enable Particle Breakage Calculations (p. 397).), then Particles Time Selection will also apply to any particle fragments (p. 1393) that were generated. Note: To avoid doubling the particle mass, parent particles from which the fragments were generated are not included in the Particles Time Selection.

If your project had breakage enabled and was processed with a previous version of Rocky, Particles Time Selection will result in no output as this feature was not compatible with breakage at the time it was processed. To resolve this, re-process your project with Rocky v4 or later.

See the image and table below to help you understand more about the Particles Time Selection User Process.

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

Figure 5.157: Particles Time Selection User Process applied to the Particles entity

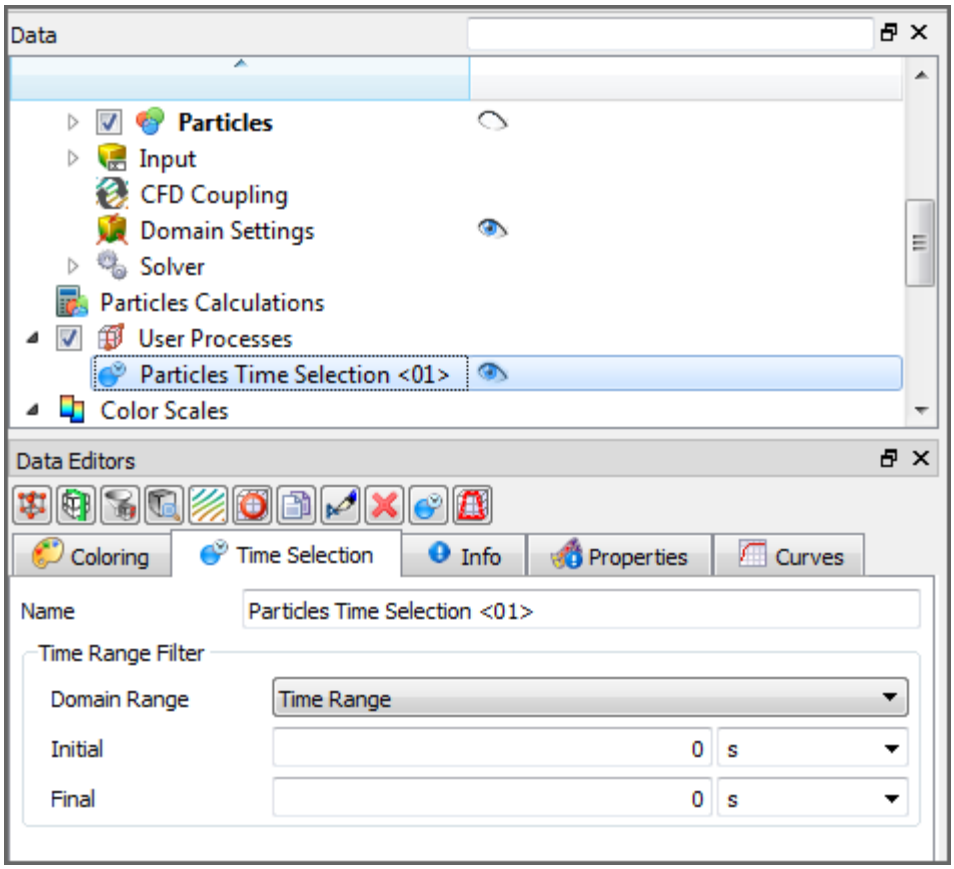


Table 1: Particles Time Selection User Process parameter options

Setting	Description	Range**
Name	Enables you to specify a unique identifier for the User Process. **Note:** This same name will appear in the Data panel under User Processes.	No limit
Time Range Filter		
Domain Range	Defines what time steps of the simulation are included in the selection. The options are as follows: • **Application Time Filter**: Use the range settings on the Time Filter dialog to limit the time steps used. (For more information, see About the Time Toolbar (p. 1279)). • **All**: Use all available time steps in the simulation.	Application Time Filter; All; Last Output; Time Range; Specify Time; After Time; Time Range Relative to Simulation End

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

Setting	Description	Range**
	• **Last Output**: Use only the very last (most recent) time step that was calculated. • **Time Range**: Limit the time step values by a specific range that you choose. (Uses the **Initial** and **Final** values you set.) • **Specific Time**: Choose only a single specific time step. (Uses the **At Time** value you set.) • **After Time**: Use all time steps available after a specific time period of your choosing. (Uses the **Initial** value you set.) • **Time Range Relative to Simulation End**: Specify how long before the very end of the simulation to include time steps. (Uses the **Range from end** value you set.)	
Initial	When **Time Range** or **After Time** is chosen for **Domain Range**, this is the starting time to begin Particles selections.	Any value between 0 and the final simulation time.
Final	When **Time Range** is chosen for **Domain Range**, this is the ending time when Particles selections are stopped.	Any value between the **Initial** time and the final simulation time.
At Time	When **Specific Time** is chosen for **Domain Range**, this is the exact moment in which the Particles will be selected.	Any value between 0 and the final simulation time.
Range from end	When **Time Range Relative to Simulation End** is chosen for **Domain Range**, this is the period of time before the final simulation time in which time steps will be included in the selection. For example, when you want to include only the last X seconds of the simulation.	Any value between 0 and the final simulation time.

See Also:

- Analyze Data with the Particles Time Selection User Process (p. 1225)

Analyze Data with the Particles Time Selection User Process

Note:

If your project was processed in a version of Rocky prior to v4 and enabled breakage, Particles Time Selection will produce no output. To solve this, re-process your simulation in Rocky v4 or later. (See also Enable Particle Breakage Calculations (p. 397).)

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

1. From the **Data** panel, right-click the Particles entity (p. 1395) for which you want to limit by a time range. This includes Particles, any shape User Process created from the Particles entity (Cube, Cylinder, or Polyhedron), or any Property or Plane User Process created from the Particles entity. Note: This User Process does not apply to Particles Trajectories User Processes, Cell Inspector User Processes, nor Eulerian Statistics User Processes.
2. From the right-click menu, point to **Processes**, and then choose **Particles Time Selection**. A new Particles Time Selection entry appears under **User Processes** in the **Data** panel.
3. From the **Data Editors** panel, on the **Particles Time Selection** tab, enter the options you want.

Note:

For projects with breakage enabled that were processed with Rocky v4 or later, Particles Time Selection will also apply to any particle fragments (p. 1393) that were generated. However, to avoid doubling the particle mass, parent particles from which the fragments were generated are not included in the selection.

See Also:

- About the Particles Time Selection User Process (p. 1221)

5.5.12. Divide a Cube or Cylinder User Process into Many Analysis Blocks

After creating a Cube or Cylinder User Process from the Particles entity, you can choose to filter your results even further by applying Eulerian Statistics to your existing User Process. Eulerian Statistics divides your existing shape into many separate blocks that you can then analyze on a more granular level.

What would you like to do?

- Learn more About Eulerian Statistics (p. 1226)
- Divide a Cube or Cylinder User Process into Many Analysis Blocks (p. 1226)

See Also:

- Filter Views and Data with User Processes (p. 1174)

About Eulerian Statistics

When you apply Eulerian Statistics to a Cube User Process (Figure 5.158: A Cube User Process applied to Particles with a Eulerian Statistics User Process applied to the Cube with a Properties User Process applied to the Eulerian Statistics, which results in only the blocks with particles being displayed in the 3D View (p. 1228)) or Cylinder User Process (Figure 5.161: The same mill slice with Eulerian Statistics but with a Property User Process applied limiting the blocks shown to only those with a Volume Fraction higher than 0.25 (p. 1231)) (see also Analyze Data within a Defined Shape (p. 1175)), you divide that original analysis shape into separate blocks for more granular analysis (Figure 5.159: A rotating mill slice with particles (p. 1229) and Figure 5.162: The same mill with Eulerian Statistics blocks limited by Volume Fraction but now with the blocks colored by Absolute Translational Velocity (p. 1232)). You control the number of blocks by specifying how many divisions you want in each of three directions:

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

- For Cube User Processes, these divisions are width, height, and depth as corresponds to the X, Y, and Z directions local to the cube. (These directions correspond to the i, j, k unit vectors of a Cartesian coordinate system.) When looking at the colored directional handles, these are in the red, green, and blue directions respectively. (See also Use Colored Directional Handles (p. 126).)
- For Cylinder User Processes, these divisions correspond to the radial, tangential, and axial cylindrical coordinates.

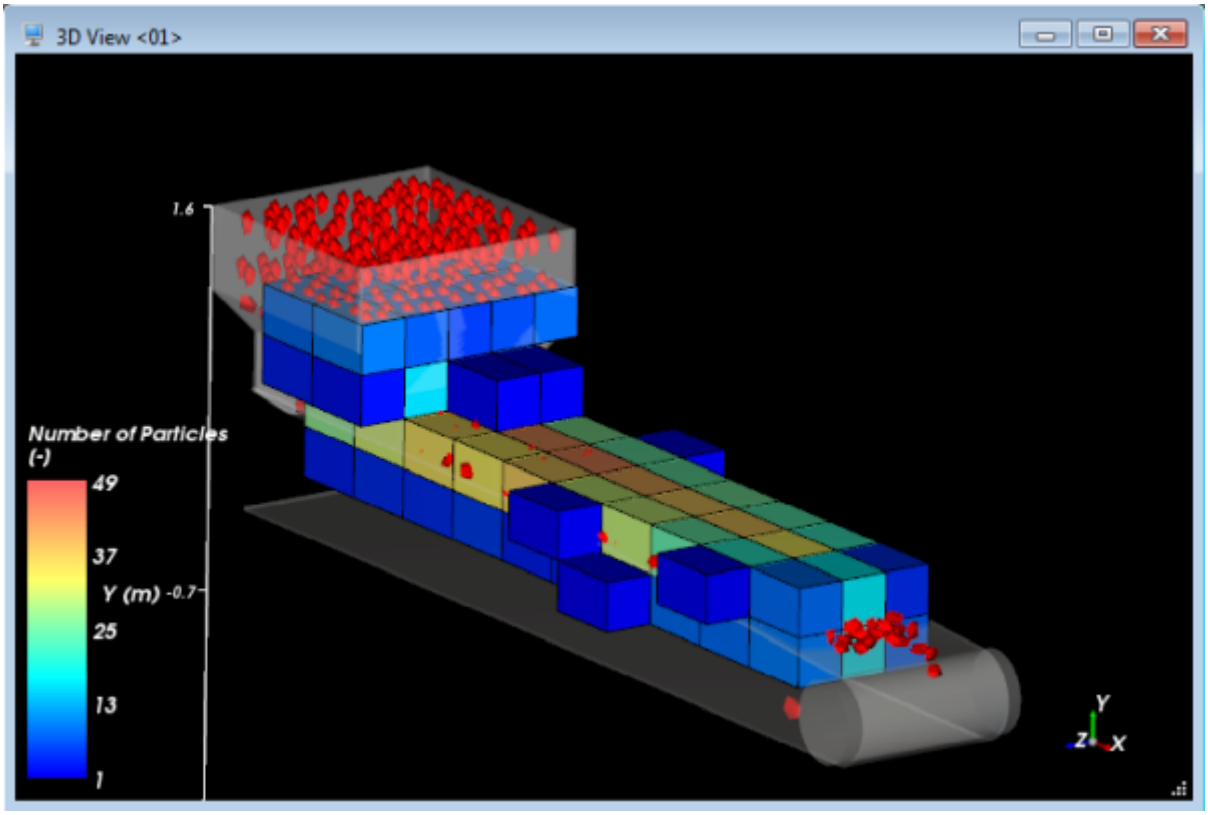
Important:

Though the edges of larger-sized cylinder blocks appear straight in the 3D View (Figure 5.158: A Cube User Process applied to Particles with a Eulerian Statistics User Process applied to the Cube with a Properties User Process applied to the Eulerian Statistics, which results in only the blocks with particles being displayed in the 3D View (p. 1228)), this is just a visualization limitation: the underlying calculations allow for proper curves. The straight-edged visualization will happen if the number of tangential divisions is too small for the curve visualization to follow. If you increase the number of tangential divisions, you will get a better-represented cylinder.

Eulerian Statistics apply only to Cube or Cylinder User Processes created from the Particles or SPH entities, but can themselves have further User Processes applied to them. For example, if you apply a Property User Process (see also About the Property User Process (p. 1202)) to a Eulerian Statistics selection, chose **Number of Particles** as the Property and set the range to be 1 or more particles, you can display only the Eulerian blocks that have particles within them.

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

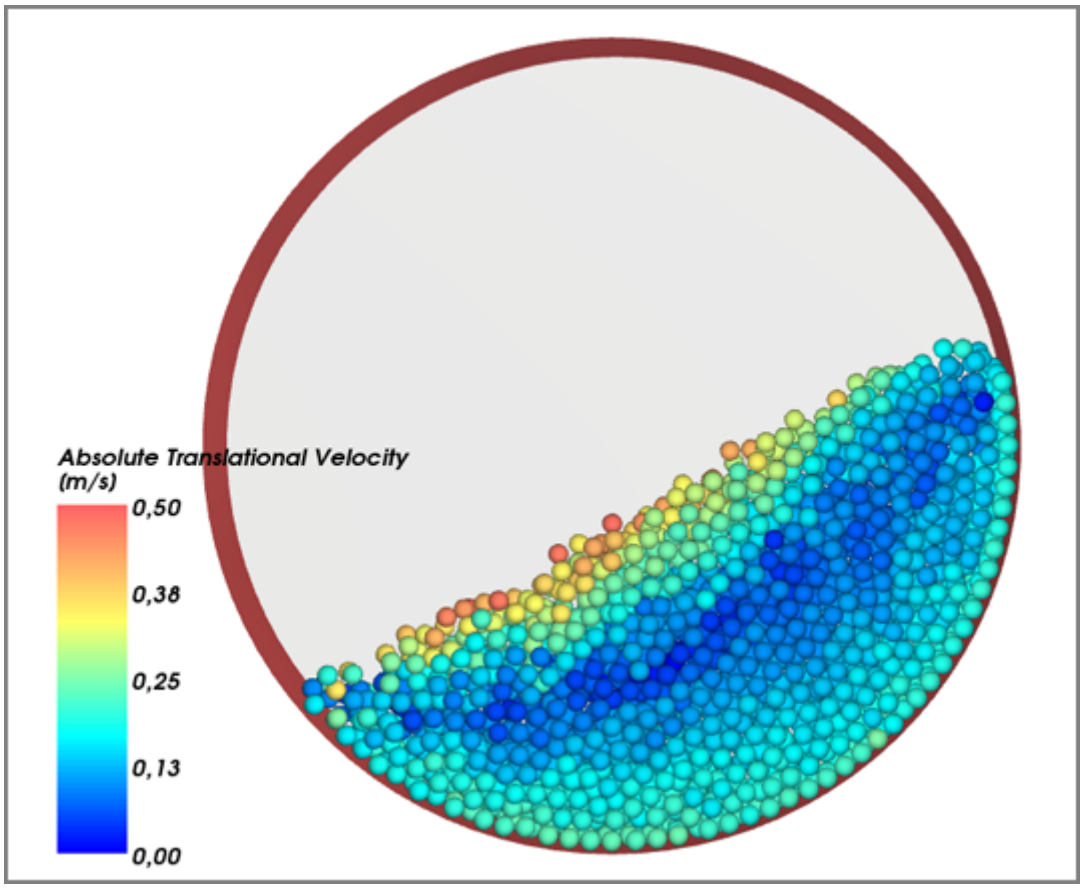
Figure 5.158: A Cube User Process applied to Particles with a Eulerian Statistics User Process applied to the Cube with a Properties User Process applied to the Eulerian Statistics, which results in only the blocks with particles being displayed in the 3D View



Another example can be seen by applying a Cylinder User Process to a mill slice, applying a Eulerian Statistics User Process to the Cylinder, and then coloring the Eulerian blocks by Particle Volume Fraction. If you then apply a Property User Process over the Eulerian Statistics User Process and limit Particle Volume Fraction to show only values above 0.25, only the blocks with those properties will be shown (see figures below for examples). Using the Coloring tab, you can then color the blocks by Average Translational Velocity, which provides a useful visualization of the bed surface.

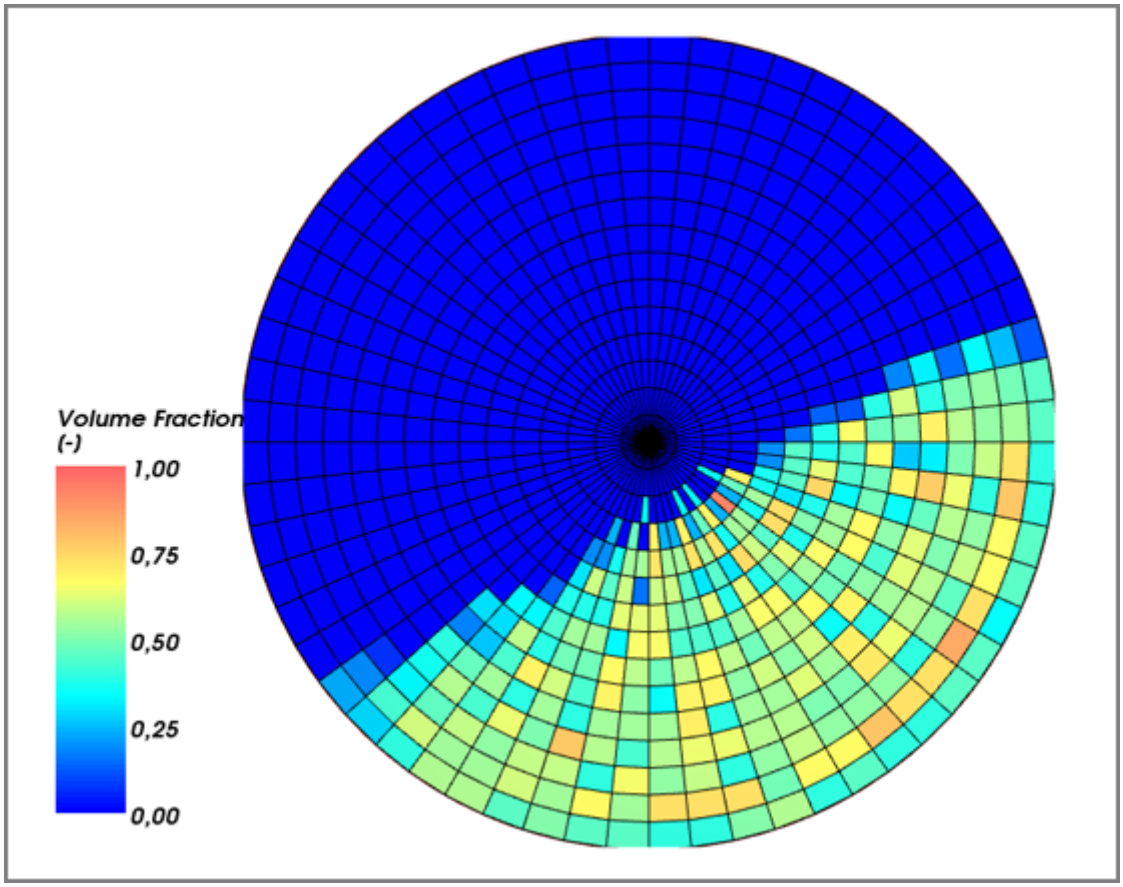
Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

Figure 5.159: A rotating mill slice with particles



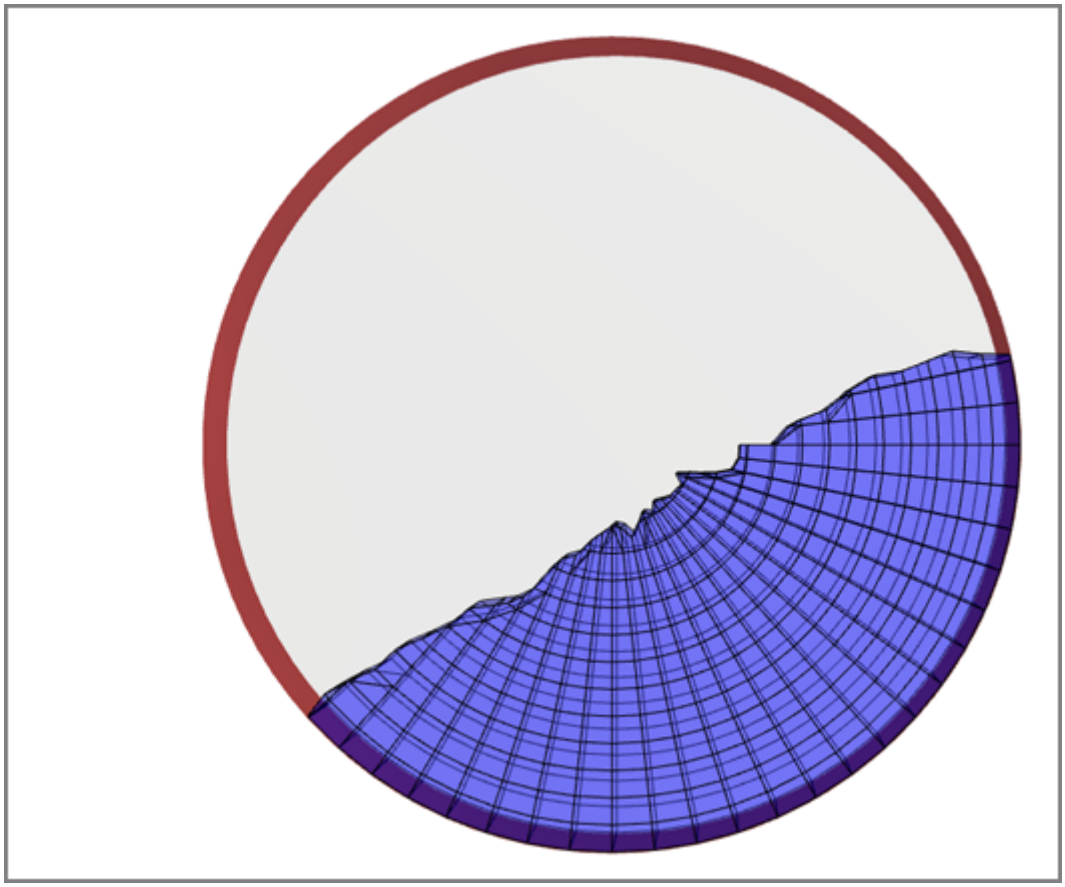
Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information
of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

Figure 5.160: The same mill slice as above but with Eulerian Statistics applied showing Volume Fraction



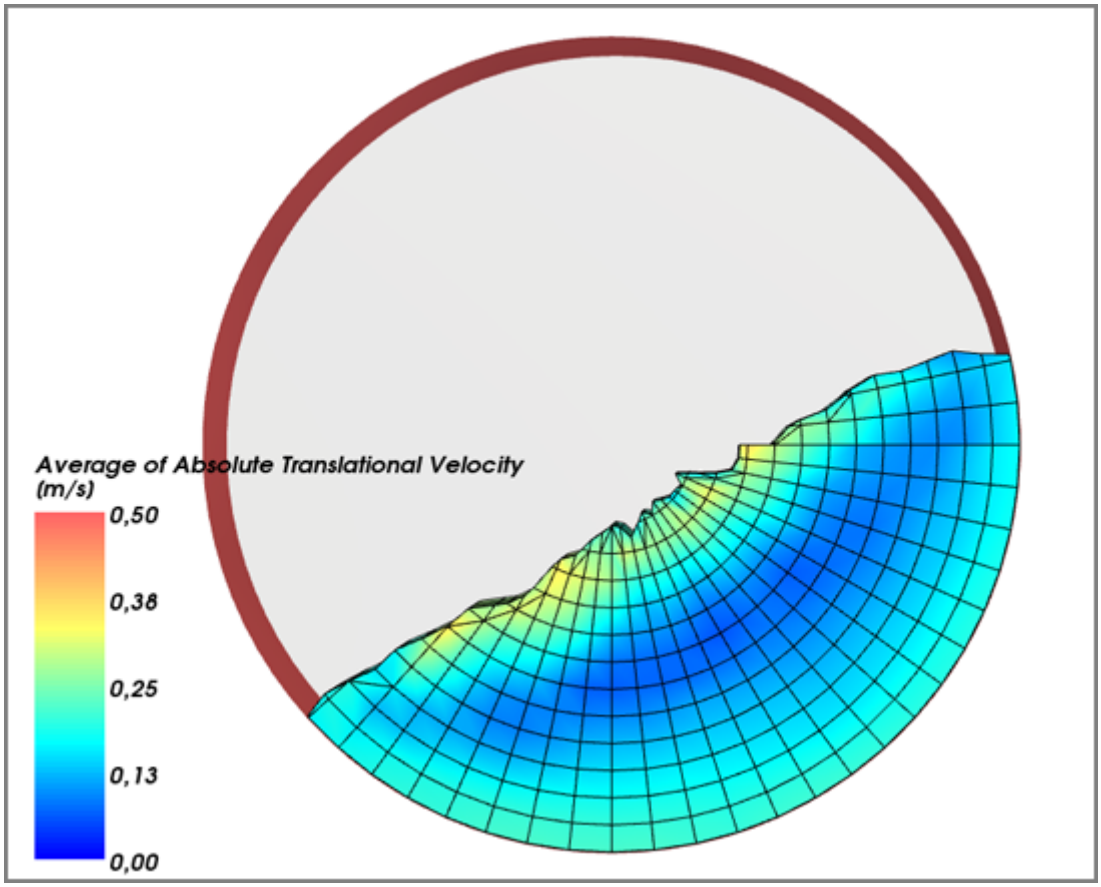
Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information
of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

Figure 5.161: The same mill slice with Eulerian Statistics but with a Property User Process applied limiting the blocks shown to only those with a Volume Fraction higher than 0.25



Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

Figure 5.162: The same mill with Eulerian Statistics blocks limited by Volume Fraction but now with the blocks colored by Absolute Translational Velocity



As with all User Processes, any new User Process that is created from another User Process is considered to be linked to the original one and will display only the chosen subset of information. (See also See Input/Output Relationships in the Data Panel When Component Names are Bold (p. 114)). This includes the unique Properties that apply only to Eulerian Statistics (see also About Properties (p. 933)).

It is also possible to inspect individual blocks using the Cell Inspector. You can also change how a Eulerian Statistics User Process appears in a 3D View by using the Coloring tab.

Lastly, another example of Eulerian User Process use is the calculation solid and liquid mass fractions.

To do so, you can add Properties to the Eulerian User Process that are not available by default. Aside from that you can add Properties from another simulation entity. For example, if your Eulerian Process was created from Particles, you can add Properties from the SPH entity. This allows you to make comparisons, such as the one below.

This example is based upon a Slurry Mill simulation. It starts by creating a Cylinder User Process over Particles, and then an Eulerian User Process upon the Cylinder.

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

Figure 5.163: A Slurry Mill with DEM Particles and SPH Elements

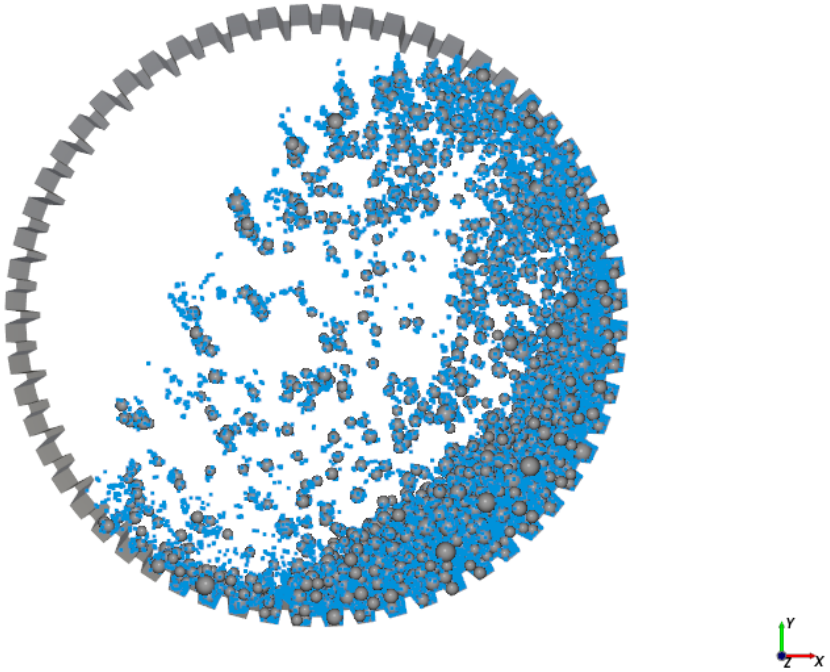
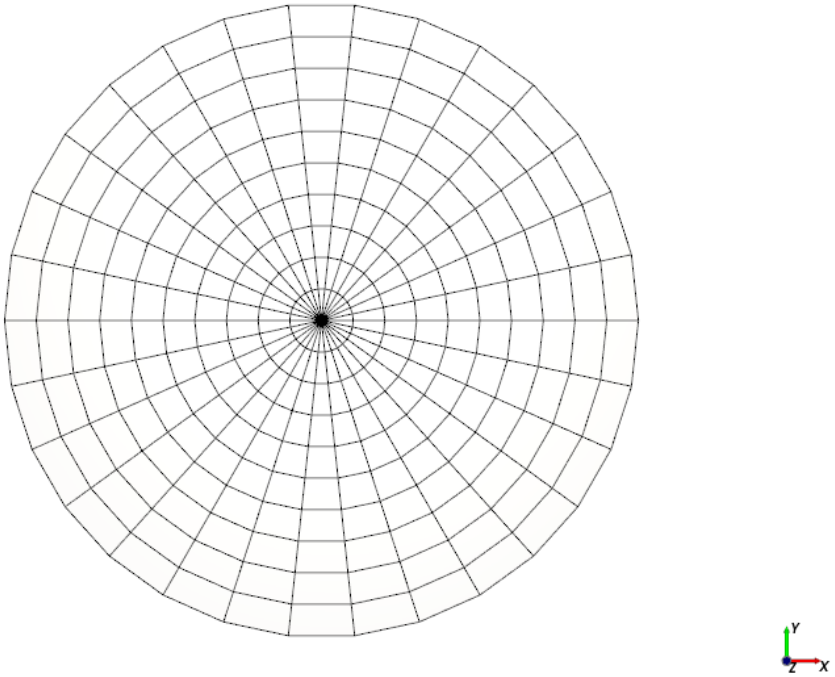


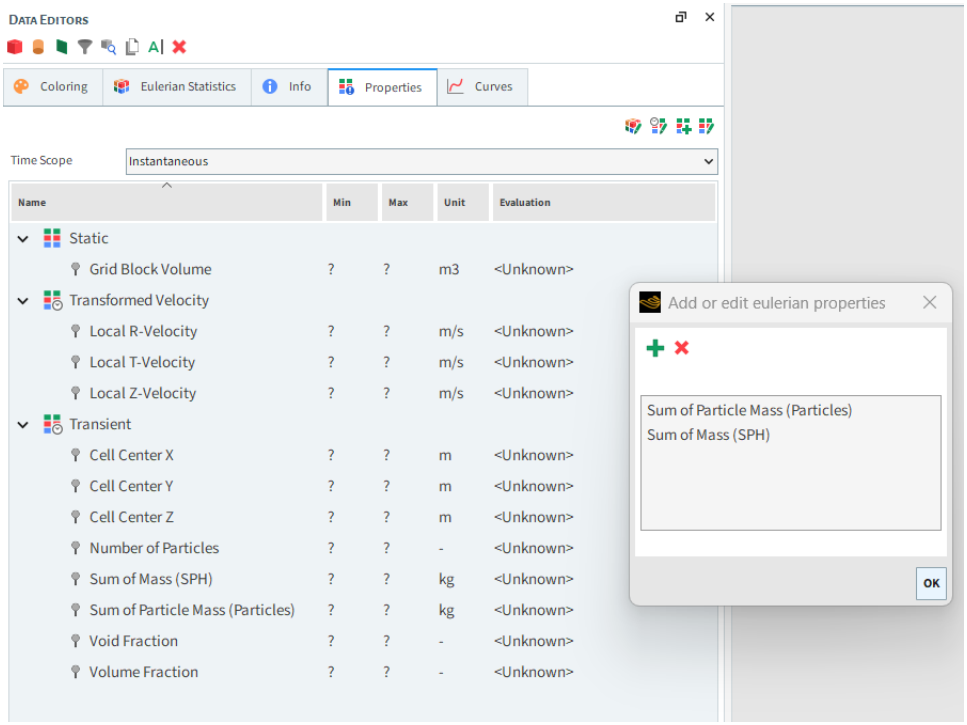
Figure 5.164: Eulerian User Process created from a Cylinder User Process



Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

Next, from the Properties tab of the Eulerian User Process, the "Add or Edit Eulerian Properties" option is used to add both Sum of Particle Mass (Particles) and Sum of Mass (SPH) properties.

Figure 5.165: Adding Properties to the Eulerian User Process

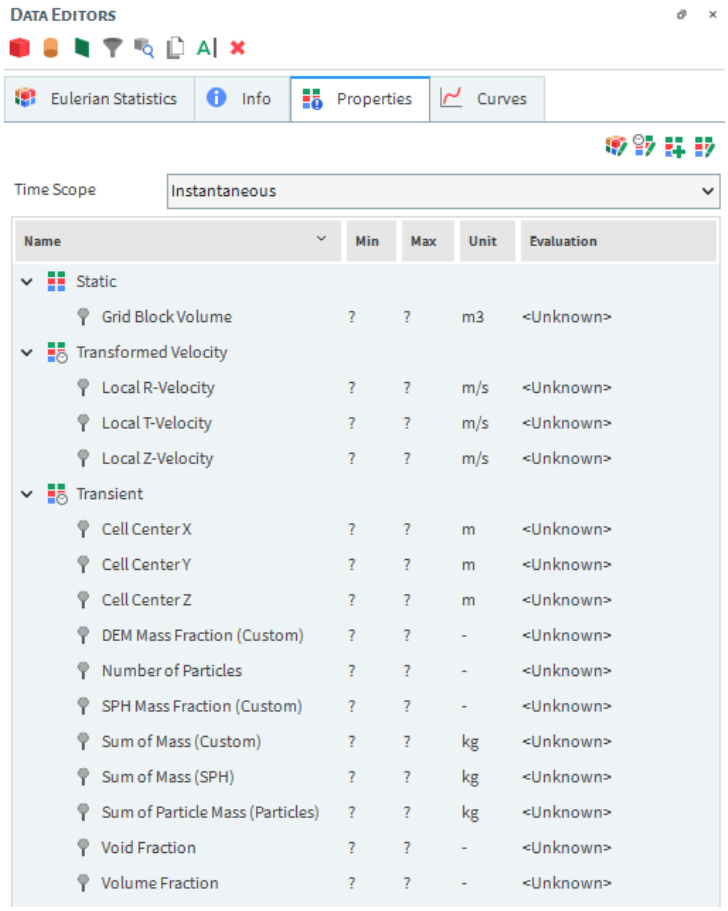


Now that the necessary Properties are available, they are used to create the "Sum of Mass" Custom Property, which is the sum of DEM and SPH mass.

From then on, two different Custom Properties can be created, DEM Mass Fraction and SPH Mass Fraction, which are the division of DEM or SPH mass by the Sum of Mass.

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

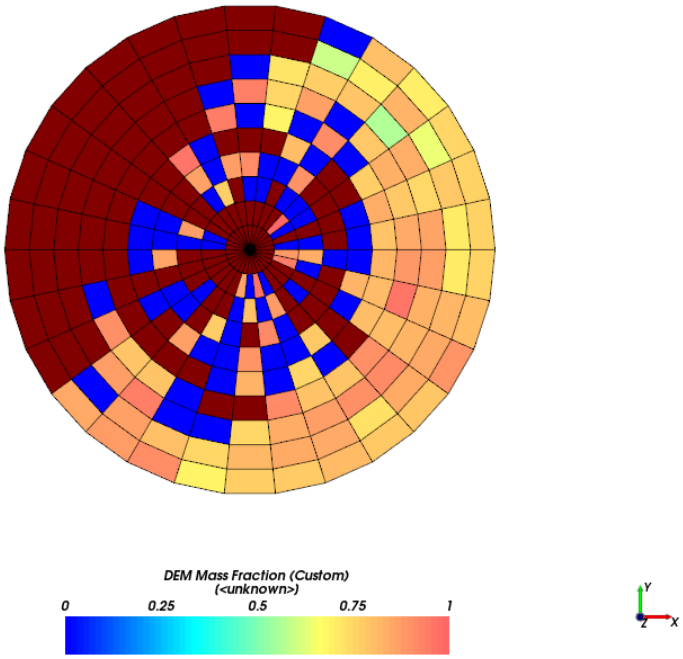
Figure 5.166: Properties tab showcasing the Custom Properties created



Lastly, the Properties that have been created are now available for Postprocessing. Below, observe the Eulerian User Process colored by DEM Mass Fraction and SPH Mass Fraction.

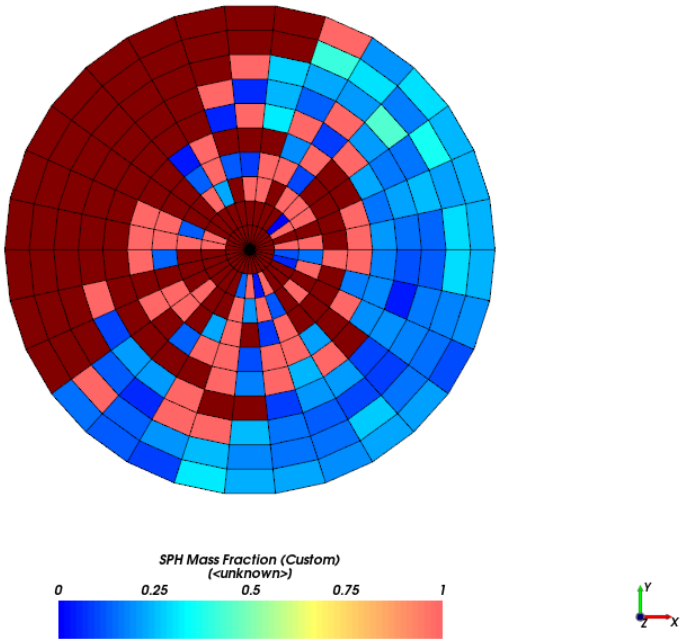
Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

Figure 5.167: Eulerian User Process colored by DEM Mass Fraction



Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information
of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

Figure 5.168: Eulerian User Process colored by SPH Mass Fraction



Important:

If you want to analyze Stress-related properties for Eulerian Statistics, you must first choose to **Collect Contacts Data** from the **Contacts** entity prior to processing your simulation. (See also About Contacts (p. 746)).

Tip:

To see walk-through examples of Eulerian Statistics, review the following Tutorials:

- Workshop 04: SAG Mill
- Workshop 09: Tablet Coater
- Workshop 18: Mixing Analysis (Liquid Bridge and LMI)

EULERIAN STATISTICS PARAMETER DEFINITIONS

See the images and table below for more information about Eulerian Statistics.

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

Figure 5.169: Cube User Process from the Particles entity

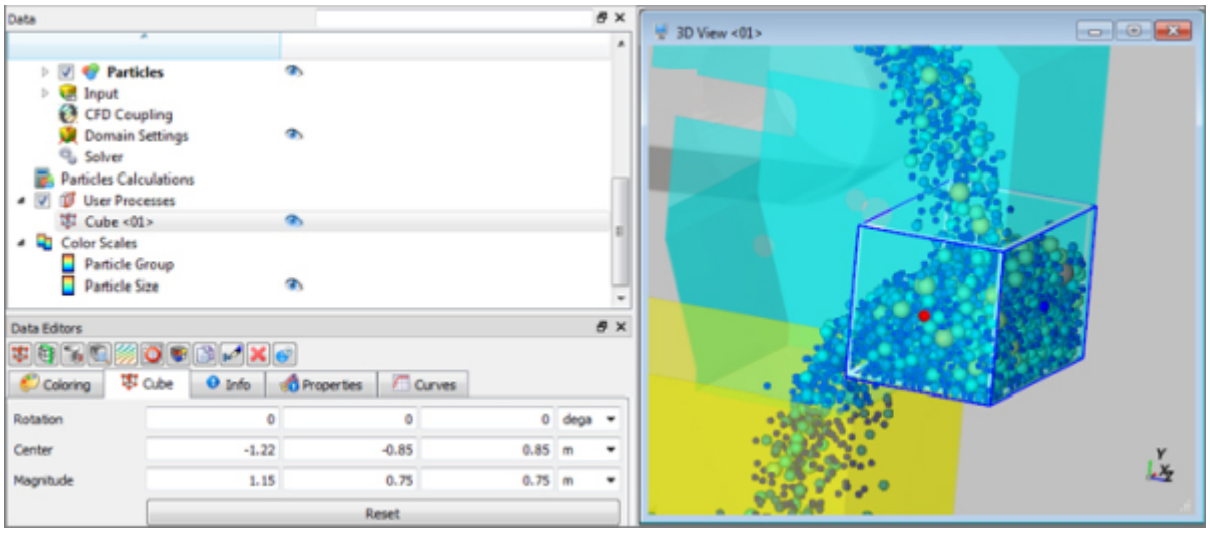
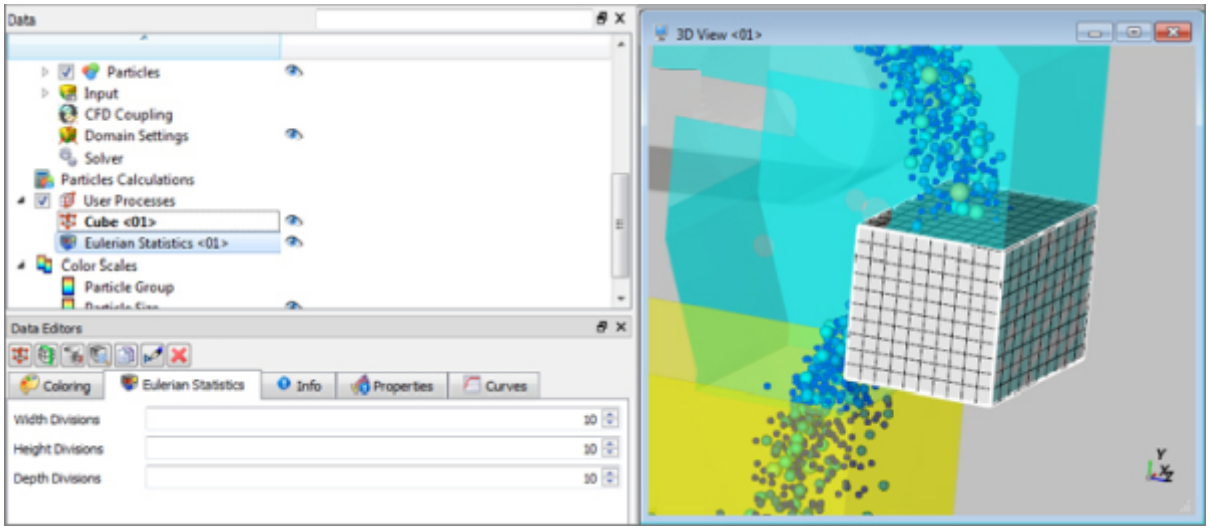


Figure 5.170: Cube User Process from the Particles entity with Eulerian Statistics applied



Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

Figure 5.171: Cube User Process showing red (local X), green (local Y) and blue (local Z) directional handles

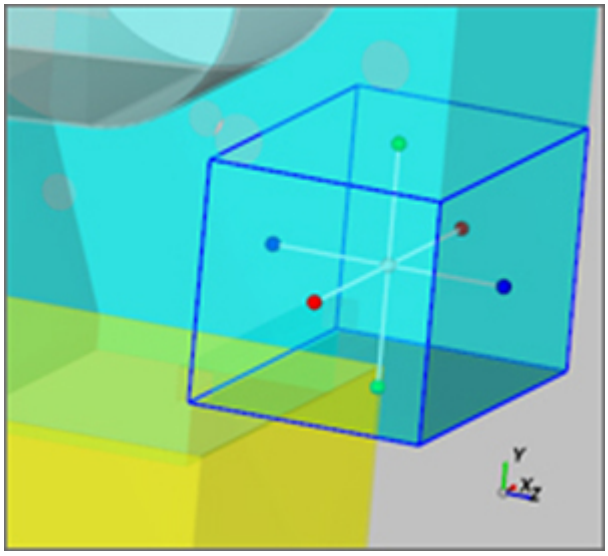


Figure 5.172: Cylinder User Process from the Particles entity

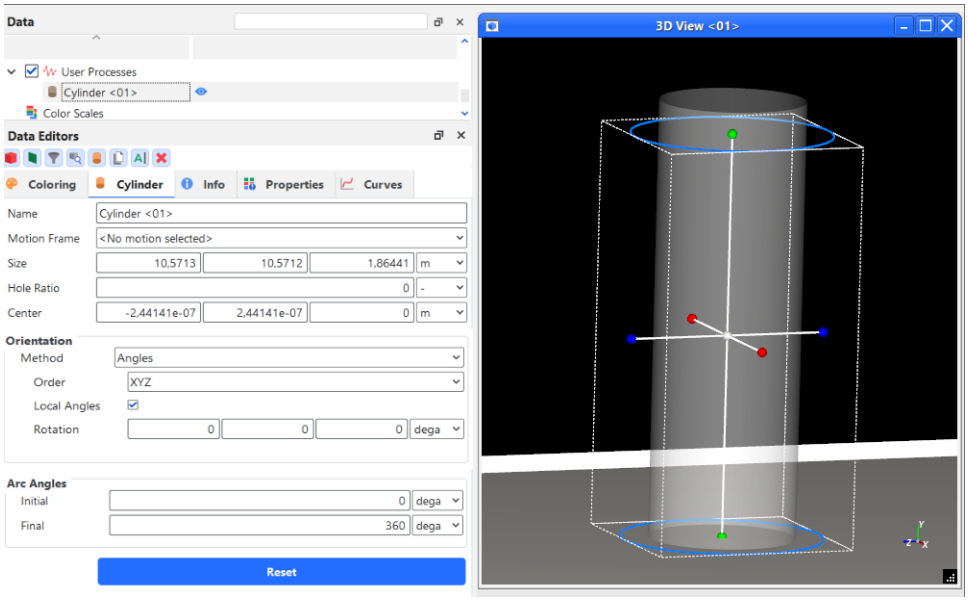


Figure 5.173: Cylinder User Process from the Particles entity with Eulerian Statistics applied

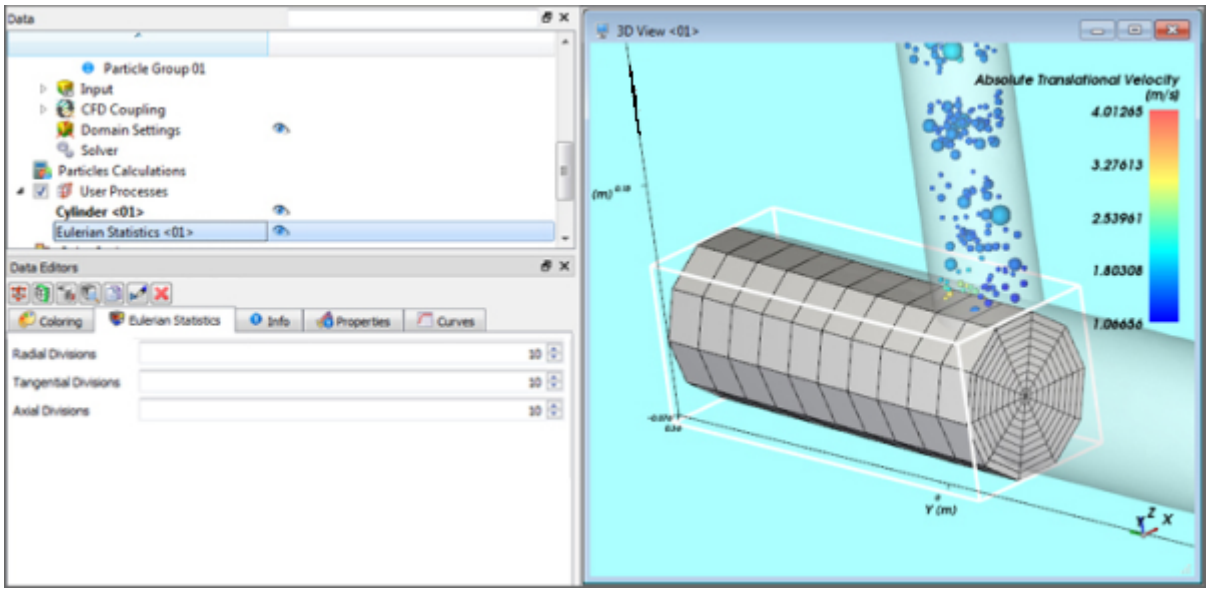


Figure 5.174: Cylinder User Process with Eulerian Statistics displaying 3 radial, 10 tangential, and 2 axial divisions

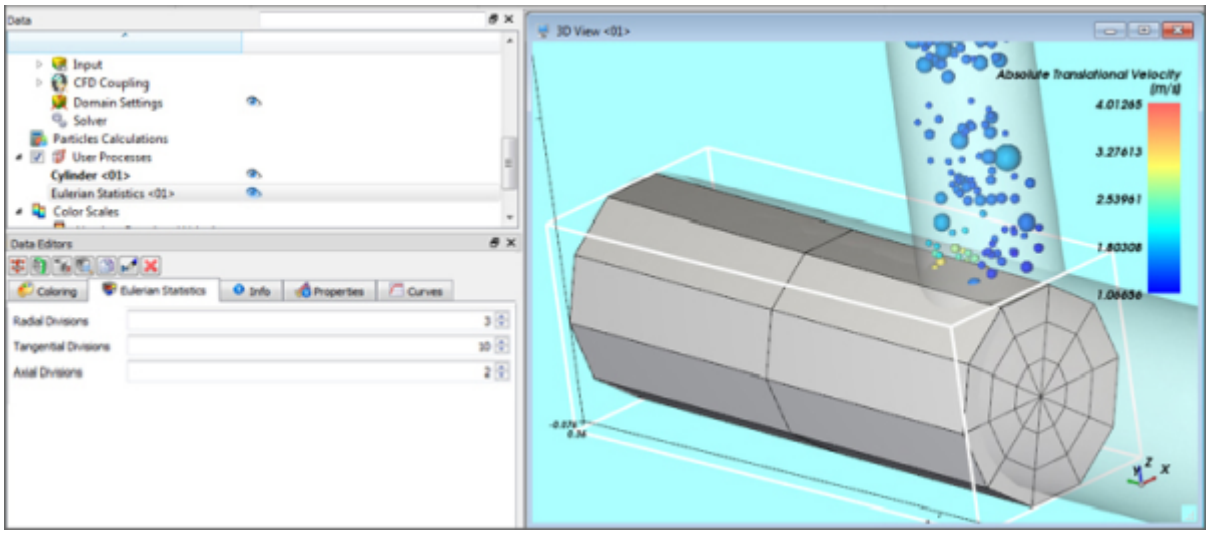


Table 1: Eulerian Statistics Options

Setting	Description	Range
Name	Enables you to specify a unique identifier for the User Process. **Note:** This same name will appear in the **Data** panel under **User Processes**.	No limit

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

Setting	Description	Range
Cube User Processes		
Width Divisions	For Cube User Processes, this is the amount of blocks you want the cube divided into in the X direction (local; corresponds to the red directional handle).	Values greater than or equal to 1
Height Divisions	For Cube User Processes, this is the amount of blocks you want the cube divided into in the Y direction (local; corresponds to the green directional handle).	Values greater than or equal to 1
Depth Divisions	For Cube User Processes, this is the amount of blocks you want the cube divided into in the Z direction (local; corresponds to the blue directional handle).	Values greater than or equal to 1
Cylinder User Processes		
Radial Divisions	For Cylinder User Processes, this is the amount of blocks you want the cylinder divided into in the X or radial direction.	Values greater than or equal to 1
Tangential Divisions	For Cylinder User Processes, this is the amount of blocks you want the cylinder divided into in the Y or tangential direction.	Values greater than or equal to 1
Axial Divisions	For Cylinder User Processes, this is the amount of blocks you want the cylinder divided into in the Z or axial direction.	Values greater than or equal to 1

What would you like to do?

- Divide a Cube or Cylinder User Process into Many Analysis Blocks (p. 1226)
- Inspect an Individual Cell (Triangle, Block, or Node) for an Entity (p. 1218)
- Learn more About Customizing Eulerian Statistics Properties (p. 1017)

See Also:

- About Properties (p. 933)
- Filter Views and Data with User Processes (p. 1174)
- About the Cell Inspector (p. 1211)
- Use Colored Directional Handles (p. 126)

Divide a Cube or Cylinder User Process into Many Analysis Blocks

1. Ensure that you have created a Cube or Cylinder User Process from the Particles entity, and have sized and positioned it the way you want. (See also Analyze Data within a Defined Shape (p. 1175).)

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

2. From the **Data** panel, under **User Processes**, right-click the Cube or Cylinder User Process you want to divide.

3. From the right-click menu, point to **Processes**, and then choose **Eulerian Statistics**. A **Eulerian Statistics** entry appears under **User Processes**. Tip:From a 3D View window (see also Create and Modify a 3D View (p. 1040)), you can see that the cube or cylinder is automatically divided into 1000 separate blocks. (10x10x10)

4. Select the **Eulerian Statistics** User Process and then from the **Data Editors** panel, enter the three **Divisions** values. (See also About Eulerian Statistics (p. 1226).)

5. Press Enter. The divisions you selected are updated in the 3D View.

See Also:

- About Eulerian Statistics (p. 1226)
- View and Save Simulation Statistics (p. 1097)
- Use Properties and Curves (p. 931)

5.5.13. Retain Particle Data Calculated within a User Process by Using Particle Calculations

User Process selections made from the Particles entity are designed to calculate data only for the particles within the selection defined. These selections, including those made by the Cube, Cylinder, and Polyhedron shapes, take a snapshot of the particles as they are in the area defined but once they leave it, the data for the exited particles is no longer tracked.

Sometimes, as when calculating particle Residence Time and Flip Count, this method is too limiting. That is why Rocky enables you to apply these and other specific functions to a selection of Particles but have the data be retained when the particles leave the area defined. This method, accomplished with Particles Calculations, enables you to focus the view while still accessing the data you need.

Using Particle Calculations, you may also tag particles within either a whole or a divided selection, and then track just those particles both forwards and backwards in your simulation.

What would you like to do?

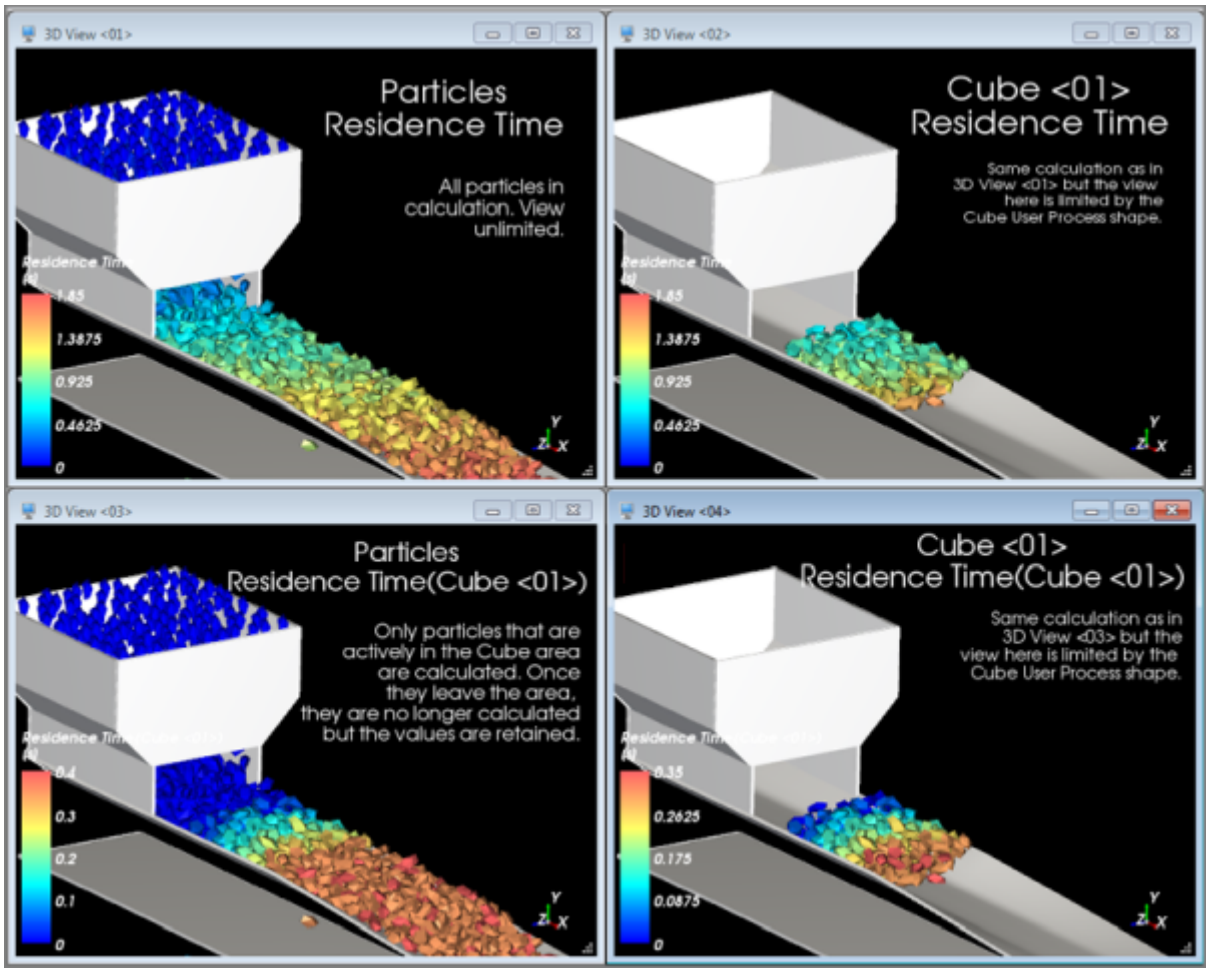
- Learn more About Particles Calculations (p. 1242)
- Learn more About Particle Tagging (p. 1248)
- Apply Particles Calculations to a Particles User Process Selection (p. 1269)
- Apply Tagging or Divisions Tagging to a Particles User Process Selection (p. 1270)

About Particles Calculations

Particles Calculations enables you to apply certain functions, including Residence Time and Flip Count (as shown below), and particle tagging (see also About Particle Tagging (p. 1248)), to only the particles in a User Process selection but retain those values after the particle leaves the selected area.

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

Figure 5.175: Residence Time applied to Particles and a Cube User Process shape respectively (top); Particles Calculations for Residence Time shown for Particles and the Cube User Process shape respectively (bottom).

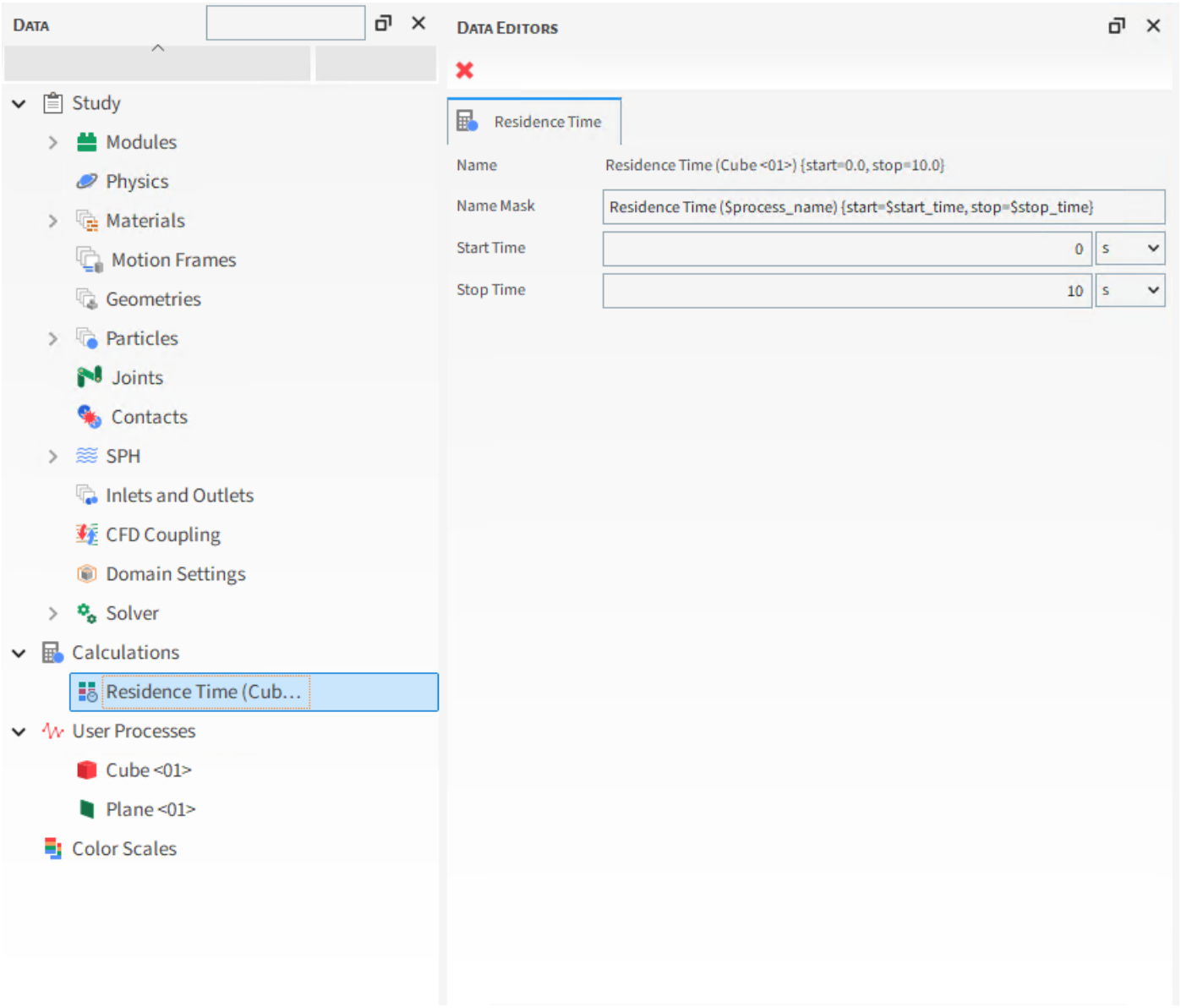


RESIDENCE TIME CALCULATION

It is possible for you to input a start time and time range for this calculation by clicking in **Residence Time** and changing the **Start** and **Stop** time in the **Data Editors**, as shown in the image below:

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

Figure 5.176: Residence Time Calculation



Where:

- **Name:** Is the unique identifier that is displayed in the Properties list and also on the Color Bar when the Property is dragged to a 3D View. By using the **Name Mask** setting, you may change the Name to whatever you want but by default, the Name is provided in the format below:

  • For Residence Time: Residence Time(Cube <01>) {start=0,0, stop=10.0}

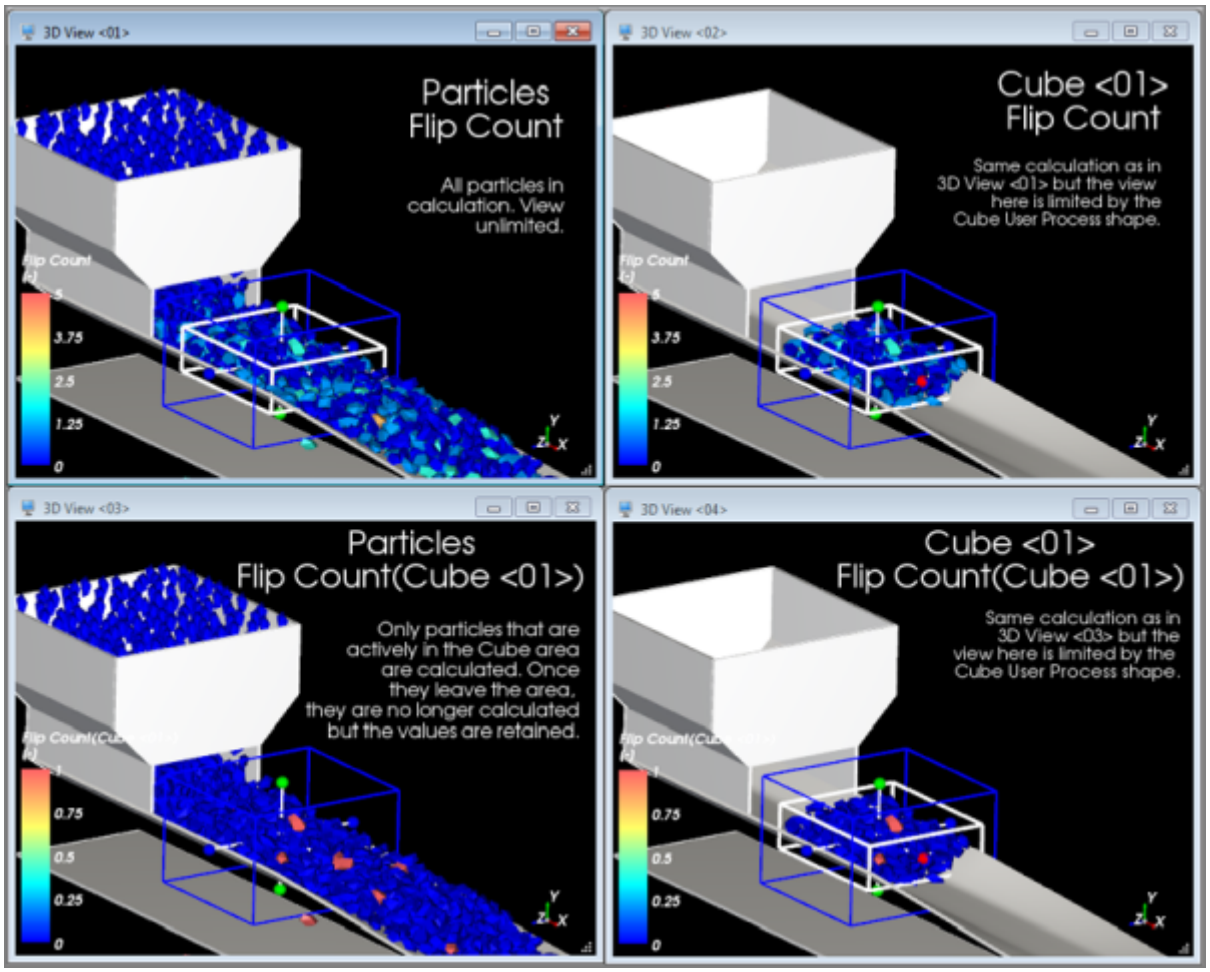
- **Name Mask:** Is the method by which you can change the **Name** using text No limit that you type or Keywords that you enter. Any Keywords you use will be replaced with the corresponding data in the Name field, and will be updated when the data supporting the Keyword changes.

- **Start Time:** The time at which you would like the residence time calculation to start.

- **Stop Time:** The time at which you would like the residence time calculation to stop.

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

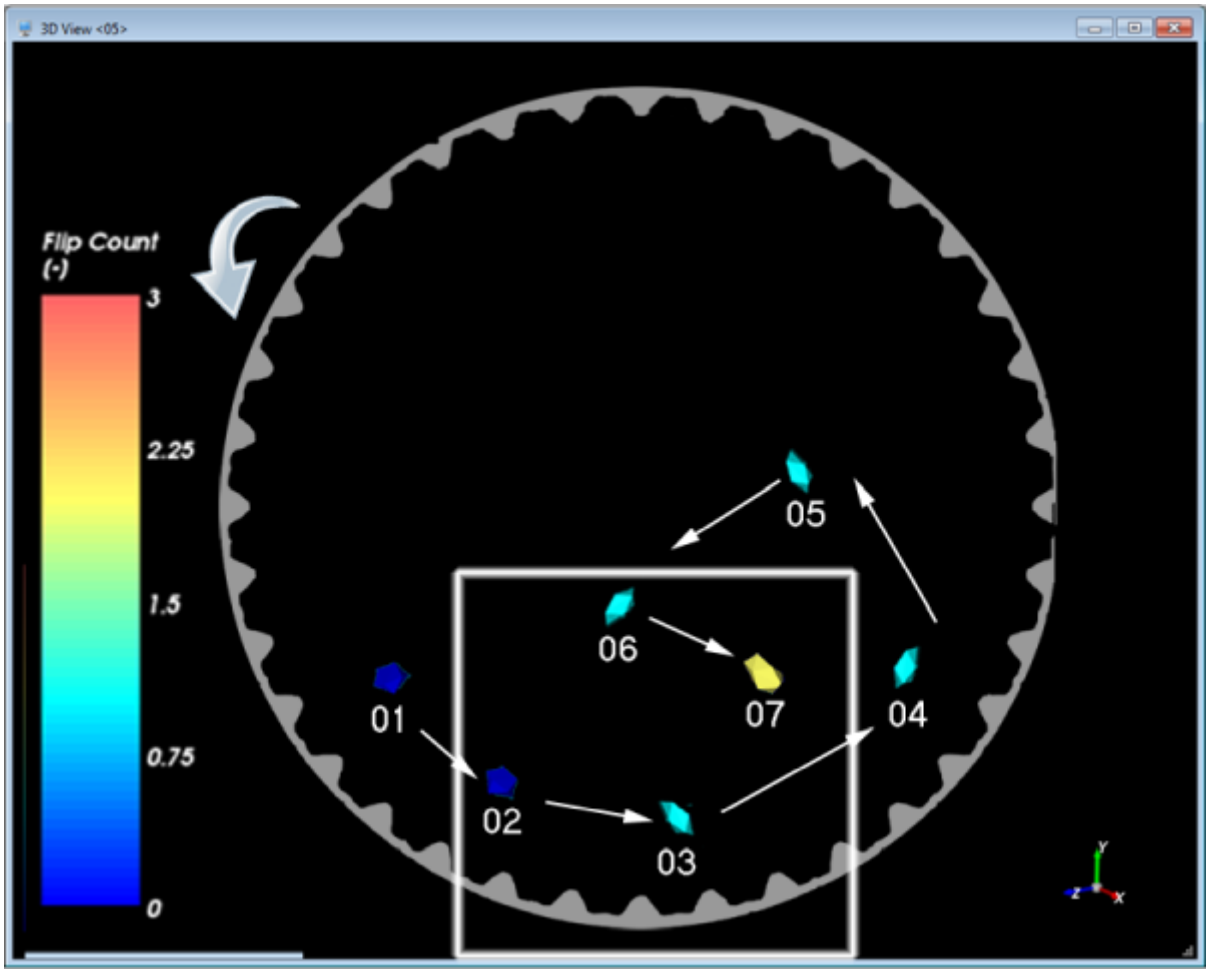
Figure 5.177: Flip Count applied to Particles and a Cube User Process shape respectively (top); Particles Calculations for Flip Count shown for Particles and the Cube User Process shape respectively (bottom).



When a particle enters the User Process area defined, the selected calculations begin. When it leaves the area, calculations stop but the value of those calculations is retained. Then, if the particle should enter the area again, calculations resume from where they left off until the particle exits the area again. This process is illustrated in the image below.

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

Figure 5.178: Mill slice illustration showing how a single particle moving through a Cube User Process is calculated for Flip Count Particles Calculations.



As illustrated in the image above, when the particle begins its journey, it is outside the cube and has no value (01). As soon as it enters the cube, calculations begin (02 & 03). As the particle exits the cube, calculations stop but the values are retained (04 & 05). When it enters the cube again, calculations resume from where they left off (06 & 07).

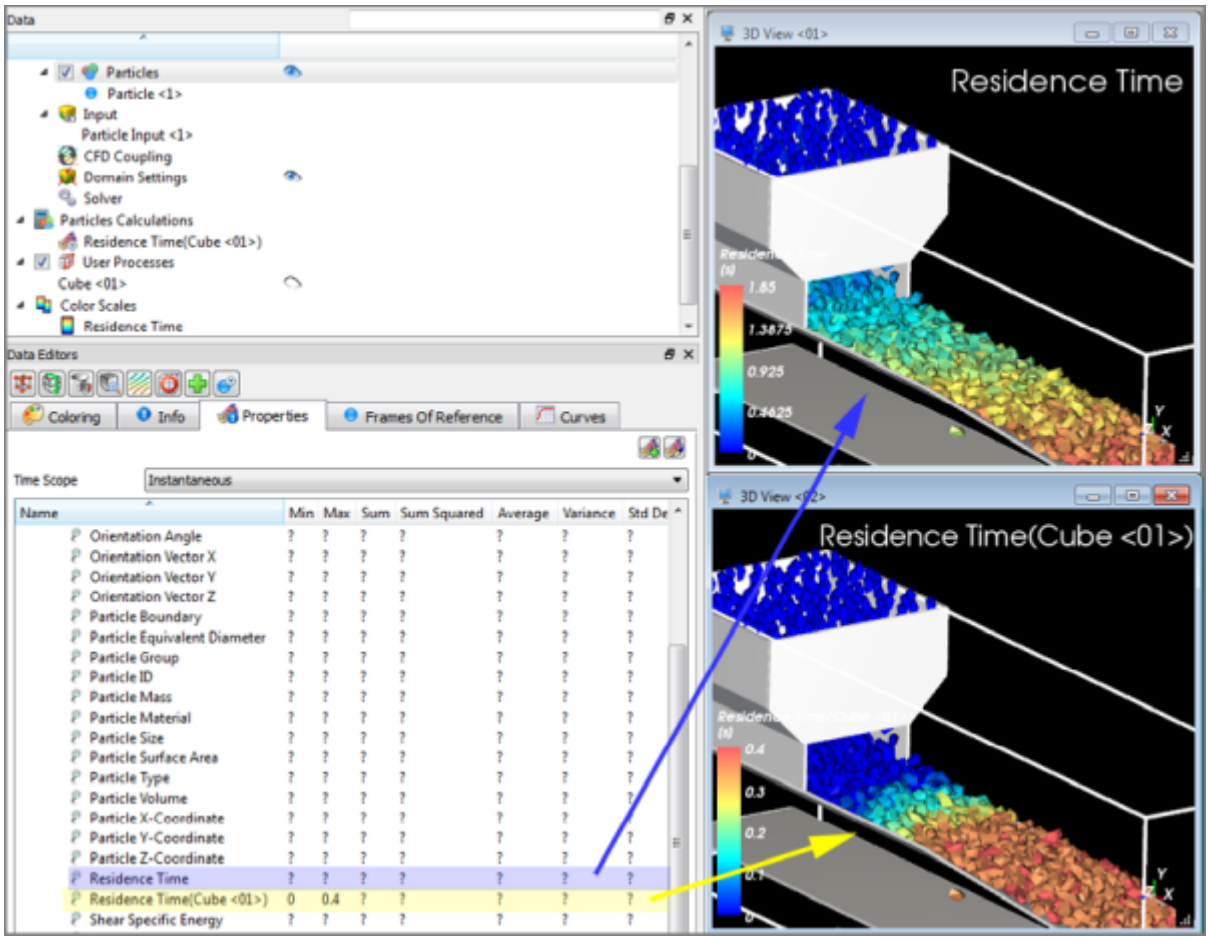
To apply Particles Calculations, you must have already defined the Particles User Process selection you want to use. The types of User Processes that support Particles Calculations includes all of the following created from either the Particles entity (p. 1395) or another Particles User Process:

- A Cube, Cylinder, or Polyhedron shape (see also Analyze Data within a Defined Shape (p. 1175))
- A Cut or Clip Plane (see also About the Plane User Process (p. 1195))
- A Property User Process (see also About the Property User Process (p. 1202))

Once you have defined the area, you apply the calculation to that area by selecting the calculation options you want from the right-click menus of either the Particles Calculation category (located on the Data panel) or the Particles entity. Once selected, a new item representing the calculation you chose appears beneath the Particles Calculation category and new properties appear in the Properties tab for both the User Process and the Particles entity (see the images below). From there, you can use the new properties just as you would any other type of property.

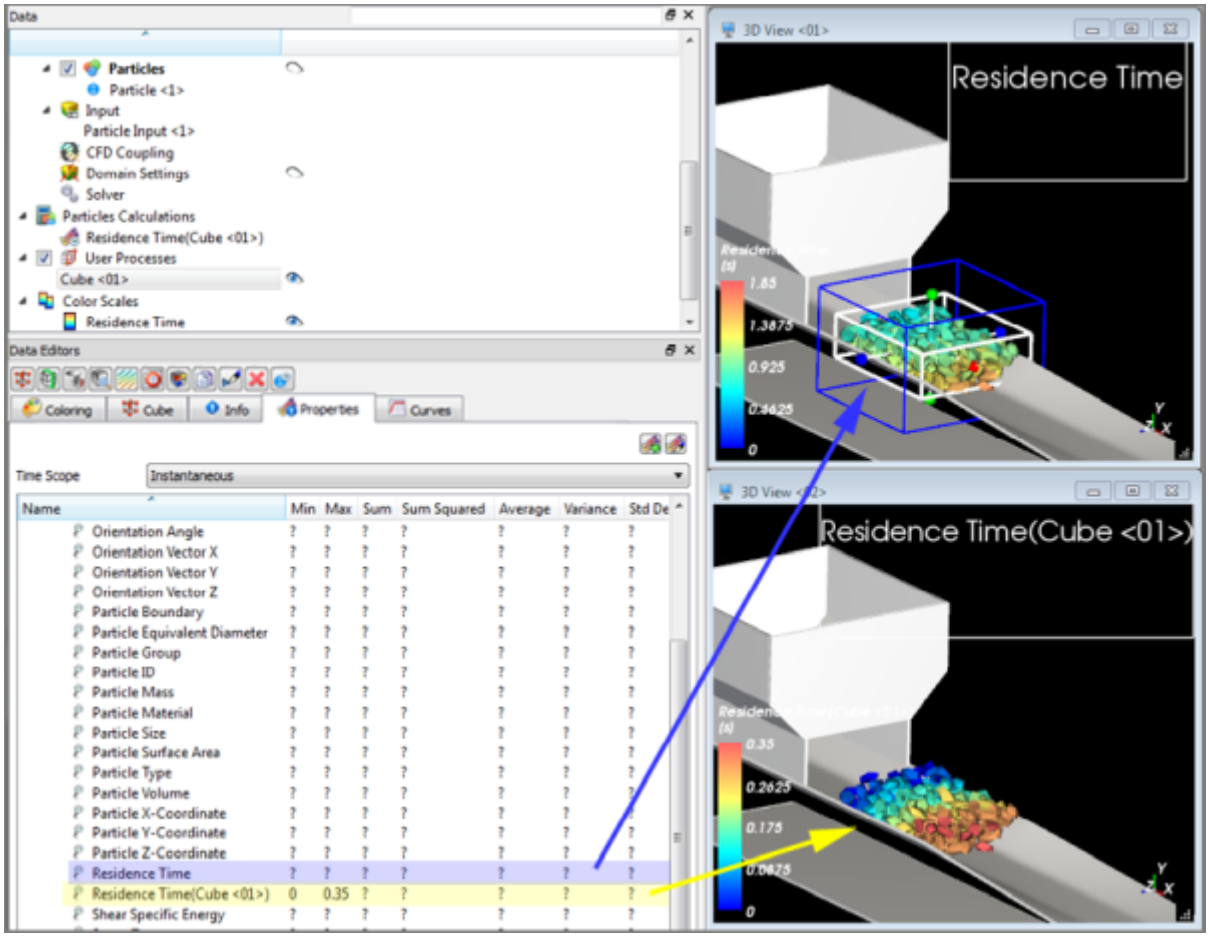
Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

Figure 5.179: Particles Calculations for Residence Time shown on the Properties tab for Particles



Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

Figure 5.180: Particles Calculations for Residence Time shown on the Properties tab for the Cube User Process



Tip:

To see a walk-through example of setting up and using a Particle Residence Time Calculation, review the Tutorial 9 in th Rocky Tutorial Guide.

What would you like to do?

- Apply Particles Calculations to a Particles User Process Selection (p. 1269)
- Learn more About Particle Tagging (p. 1248)
- Learn more About Properties (p. 933)

See Also:

- Filter Views and Data with User Processes (p. 1174)

About Tagging Calculation

Tagging Calculation, which also includes Divisions Tagging, consists in Particles or SPH Calculations (see also About Particles Calculations (p. 1242)) that you can apply to User Process selections created

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

from the Particles or SPH entity (p. 1395). When you apply Tagging Calculations those particles or SPH elements are marked with a unique identifier that you can then use to track the whole selection through the simulation–even backwards in time.

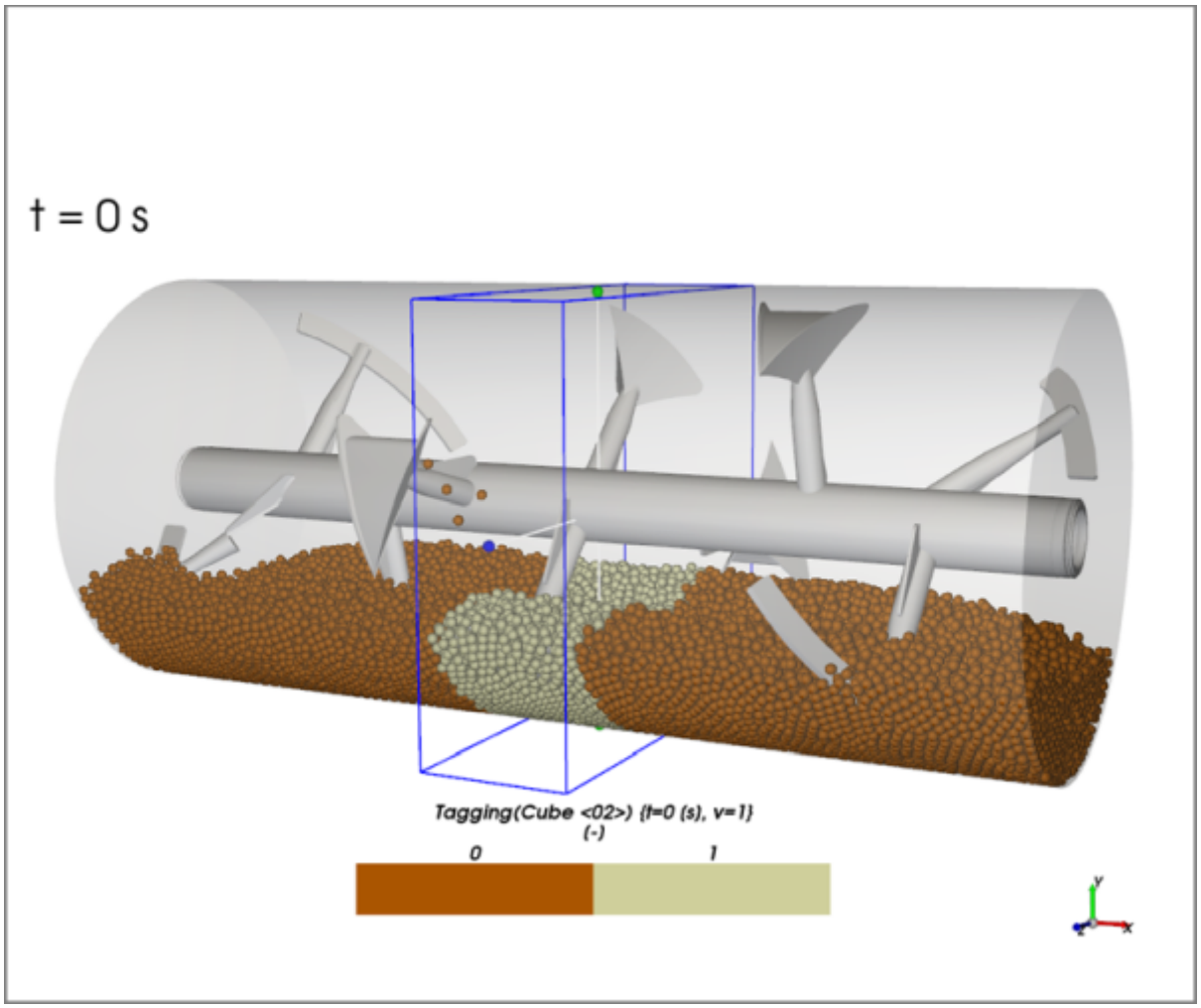
Whether breakage parameters are set and when tagging is turned on affects how this feature is applied.

In addition, Rocky offers two ways to mark your particles: Tagging and Divisions Tagging. See the sections below for further detail.

PARTICLES TAGGING

Tagging marks all particles within the User Process selection as a whole and can be used to show attributes such as mixing efficiency. In the example shown below, a mixer with shafts rotating at 30 RPM has a Cube User Process selecting only those particles in the center. To this Cube, Tagging is applied by using Particle Calculations, which then marks as "1" all particles inside the cube at the beginning of the simulation.

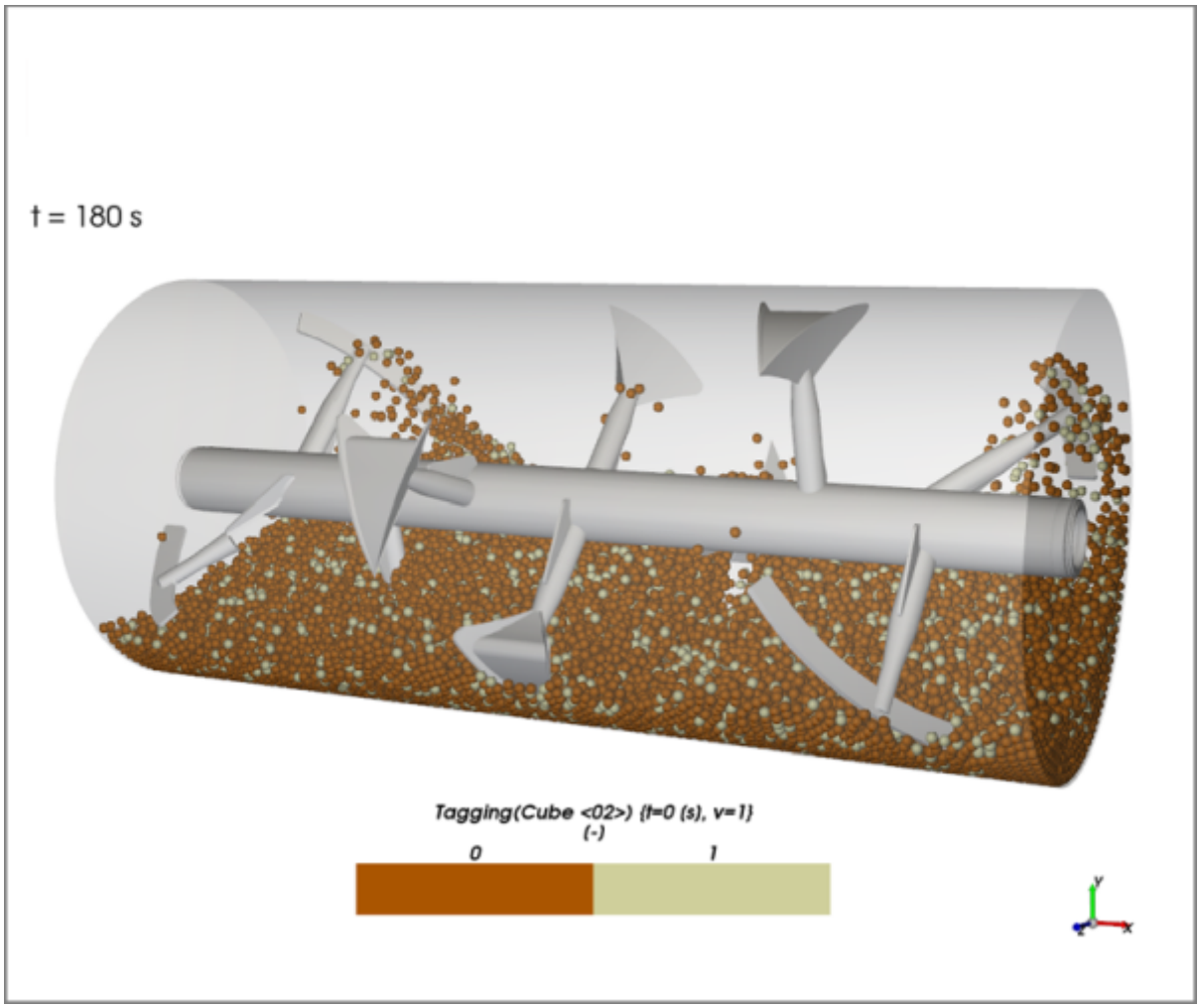
Figure 5.181: Mixing equipment at the beginning of the simulation with tagged particles in the center



As the simulation processes, the tagged particles are seen to be mixed with the non-tagged particles (as shown below).

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

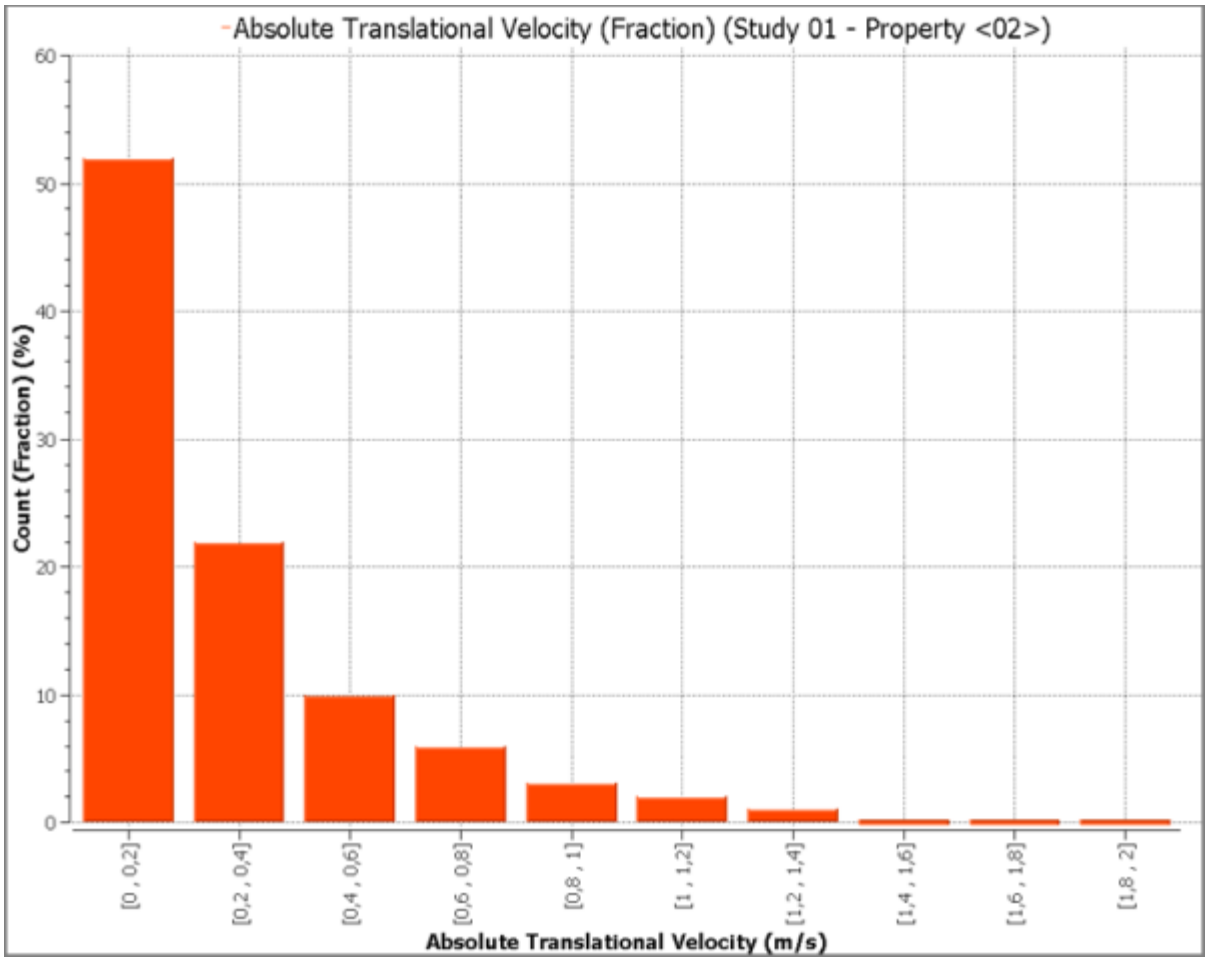
Figure 5.182: Mixing equipment later in the simulation with tagged particles mixed



Once the simulation has reached the desired level of completion, a property User Process can be applied to the Particles entity that selects only the particles which have been tagged with a "1". Using this information, a histogram of the Absolute Translational Velocity for the tagged particles can be calculated for the desired moment in time Figure 5.183: Histogram showing Absolute Translational Velocity of the tagged particles (p. 1251), which helps to illustrate how well the equipment is mixing.

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

Figure 5.183: Histogram showing Absolute Translational Velocity of the tagged particles



Like most other Particles/SPH Calculations, Tagging can apply to any Cube, Cylinder, Polyhedron, Plane, or Properties User Process that you have created from the ParticlesSPH entity. (See also Filter Views and Data with User Processes (p. 1174)) Tagging can also apply to just the ParticlesSPH entity itself if you want to select all particlesSPH elements in the simulation at a given time.

Tip:

To see a walk-through example of Tagging, review Tutorial - Ribbon Blender in the *Rocky Tutorial Guide*.

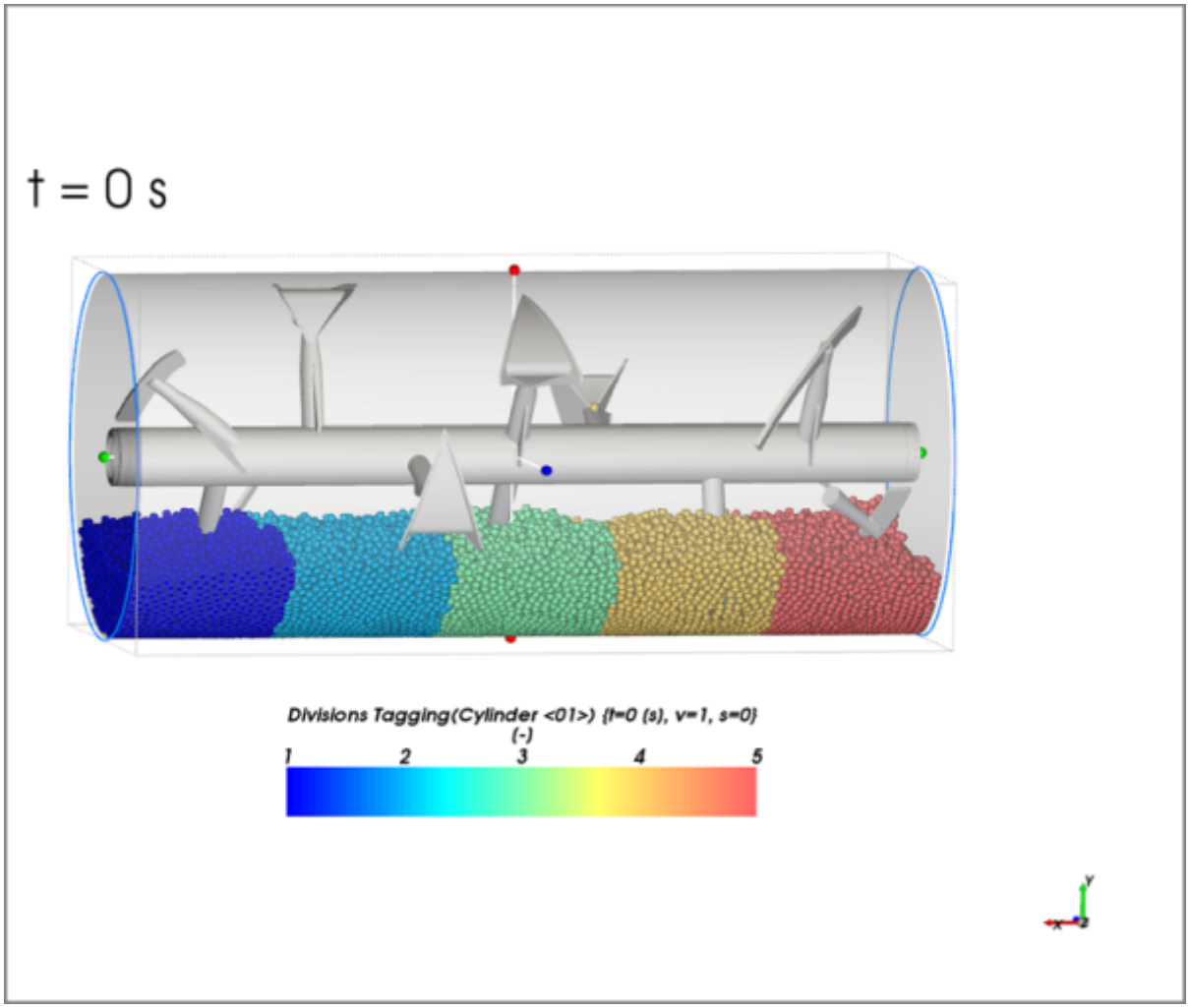
DIVISIONS TAGGING FOR PARTICLES

Regarding particles simulations, if you need a more granular analysis than offered by Tagging, you can also use Divisions Tagging, which enables you to divide your selection spatially into blocks, not unlike Eulerian Statistics (see also About Eulerian Statistics (p. 1226)). Each block can then be assigned its own unique identifier and then the particles within those blocks are tracked in the same manner as Tagging. As with Eulerian Statistics, Divisions Tagging applies only to Cube or Cylinder User Processes created from the Particles entity.

Using the same mixing example as above, the image below shows the particles split, using Divisions Tagging, into 5 groups based upon their position at the beginning of the simulation.

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

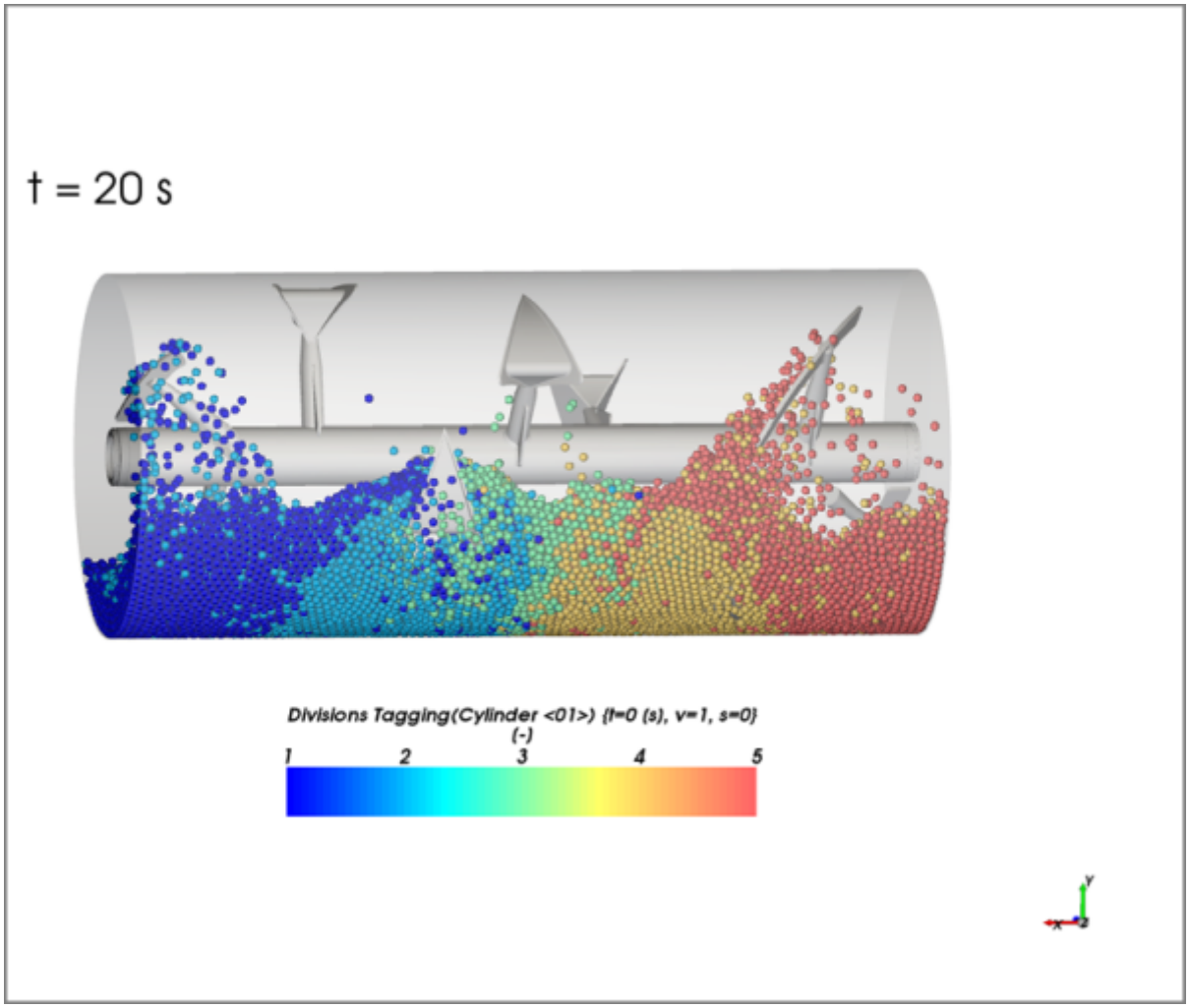
Figure 5.184: Mixing equipment with Divisions Tagging applied to create 5 separately tagged groups



As the simulation begins and the particles start to mix, the colors of the groups help to illustrate how well the particles are being mixed:

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

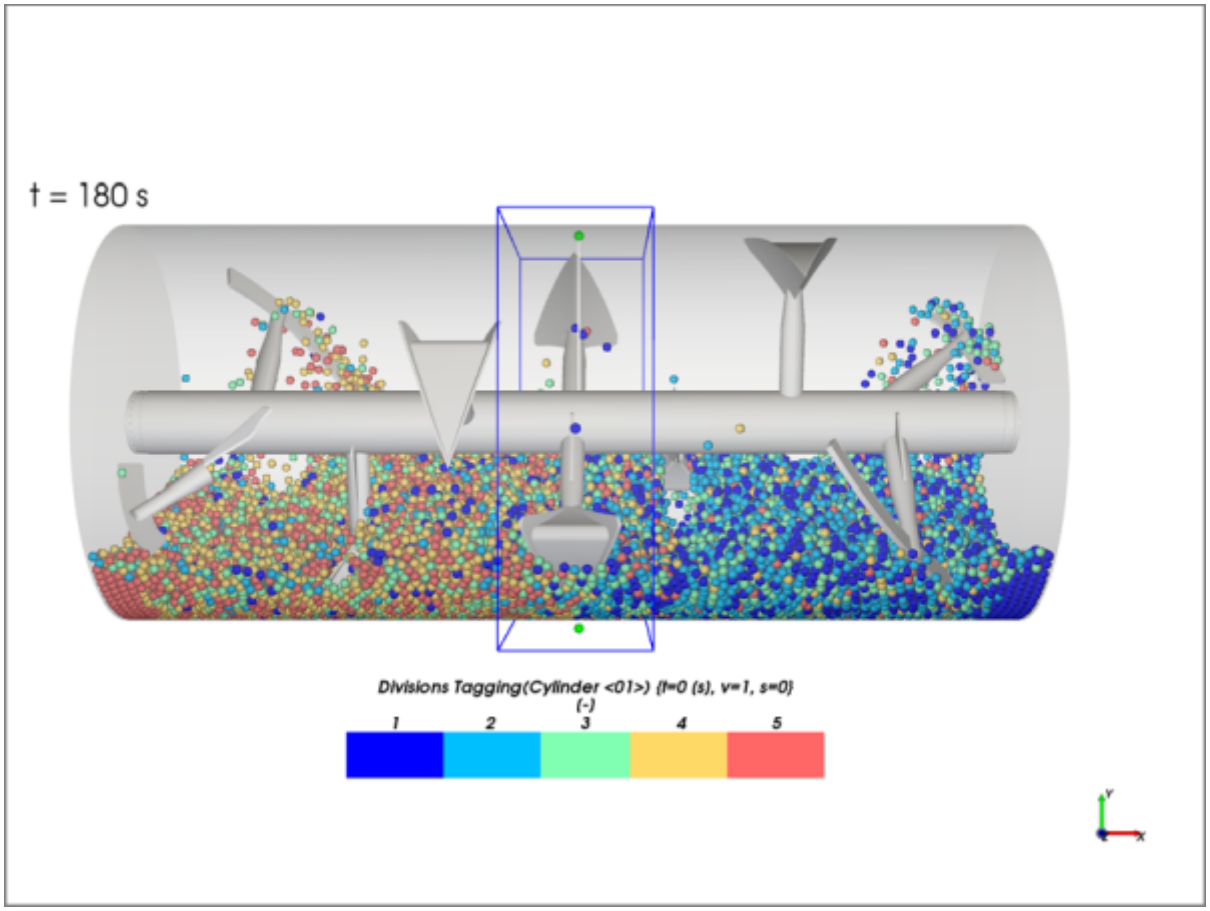
Figure 5.185: Mixing equipment later in the simulation with tagged groups of particles beginning to mix



Once the simulation reaches the desired completion, the Divisions Tagging Property can then be used for additional calculations. In the example shown above, a Cube User Process is applied to the mixed particles at 180 seconds and then a histogram of the fraction of each of the separately tagged groups is generated and compared to the initial composition (as shown below).

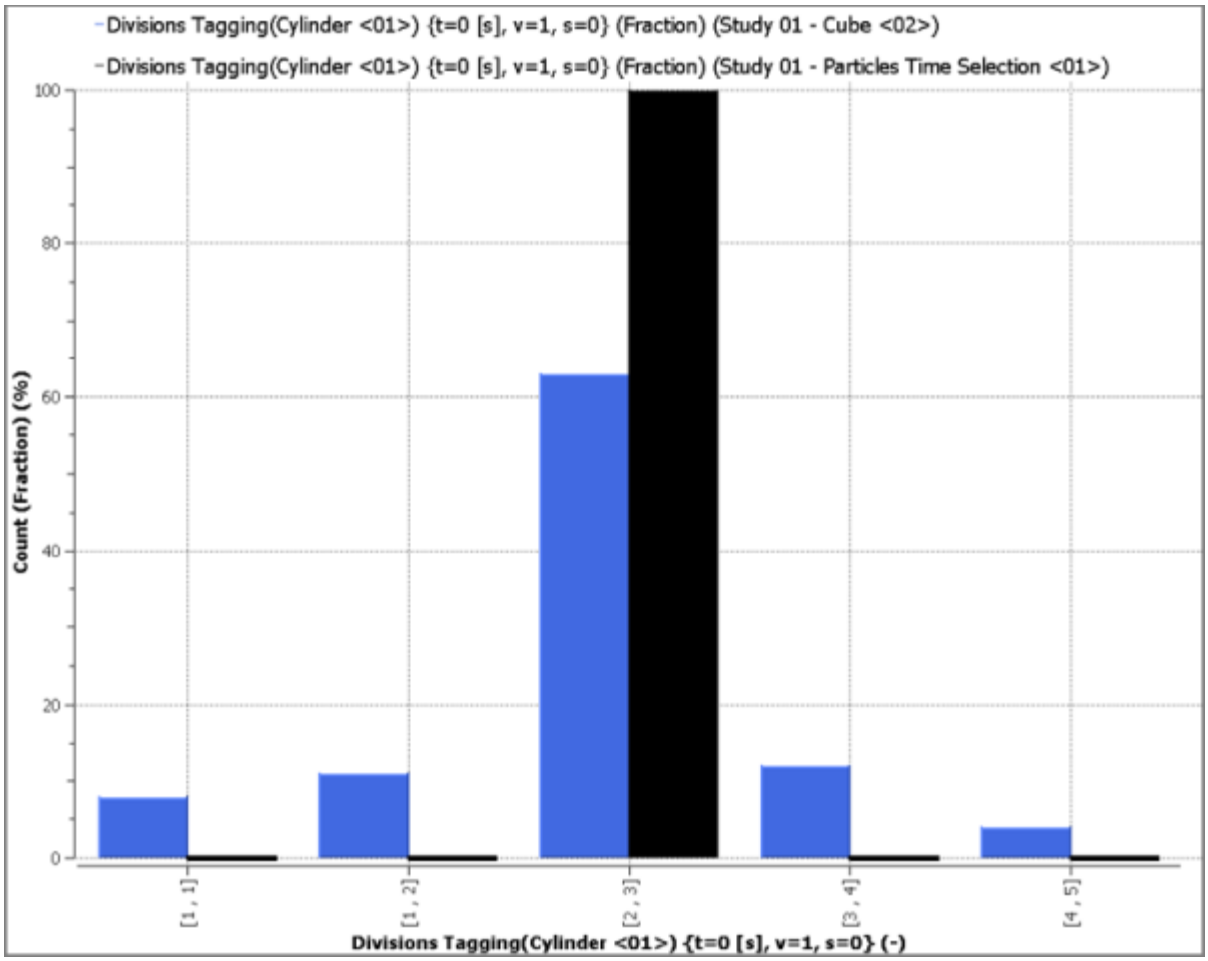
Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

Figure 5.186: Mixing equipment at the end of the simulation with a Cube User Process selection applied to the center portion



Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

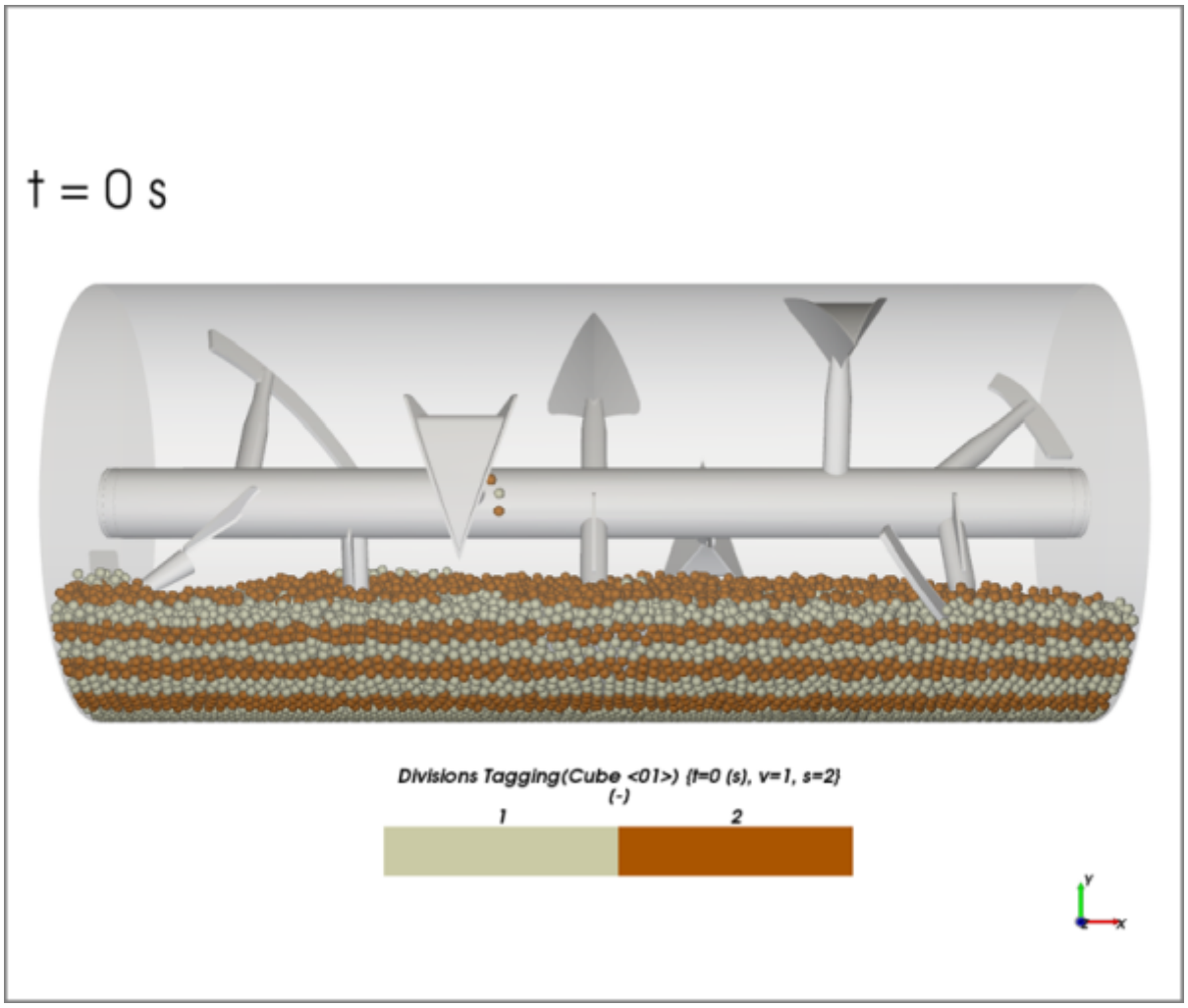
Figure 5.187: Histogram comparing a center sample of mixed particles to the initial composition



Divisions Tagging also enables you to mark your blocks in bands or striation instead of sequentially. In the example shown below, the Cube User Process encompassing the mixer is split by Divisions Tagging into horizontal rows by specifying 15 Height divisions, and 1 Depth and Width divisions, with a Tag Value of 1 and a Tag Stride of 2.

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

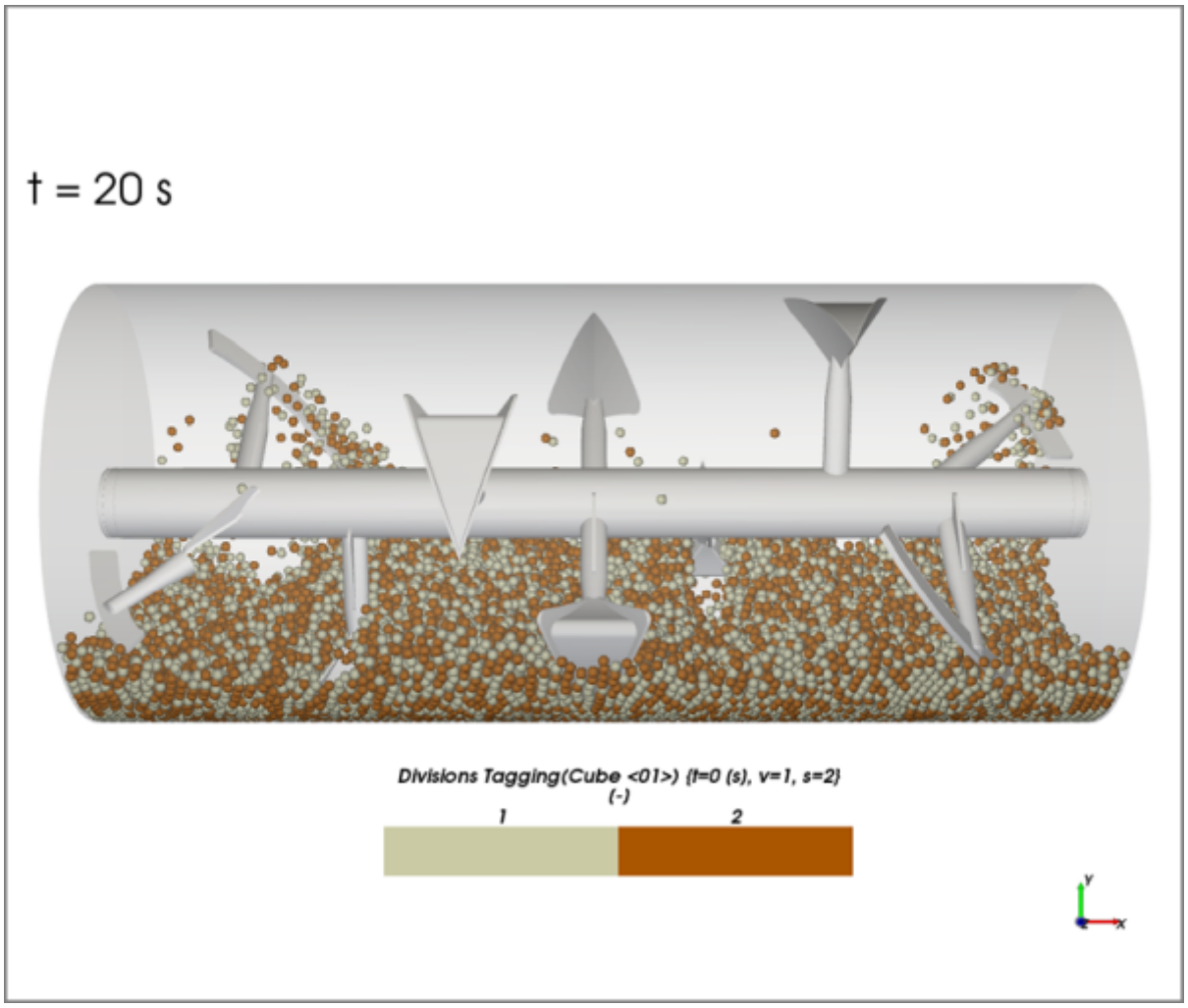
Figure 5.188: Particles with Divisions Tagging showing a Tag Stride of 2



After 20 seconds, you can see the particles colored by tagging are thoroughly mixed:

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

Figure 5.189: Particles with Divisions Tagging after being mixed for 20 seconds



Tip:

To see walk-through examples of Divisions Tagging, review the following Tutorials:

- Tutorial - Cone Crusher in the *Rocky Tutorial Guide*
- Tutorial - Fluidized Bed in the *Rocky Tutorial Guide*

TAGGING AND BREAKAGE

If you have processed your simulation and have breakage modeling enabled(see also Enable Particle Breakage Calculations (p. 397).), both Tagging and Divisions Tagging will apply to any fragments (p. 1393) that are generated. If you tag particles before they break, the fragments will inherit the tag value of

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

their parent particle. If you tag particles after they have been broken, each fragment is treated individually. (This applies to both Tagging and Divisions Tagging.)

Note:

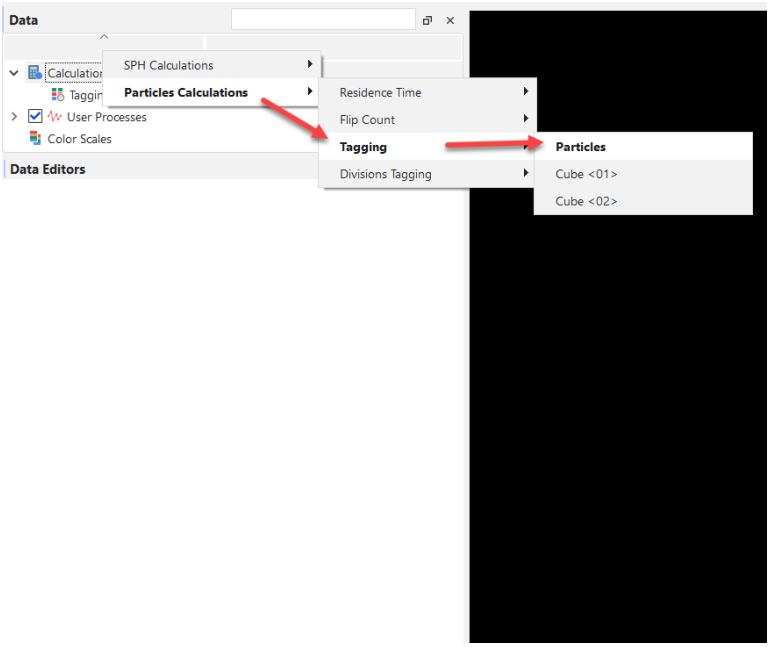
Neither Tagging nor Divisions Tagging are compatible with breakage simulations that have been processed with versions of Rocky prior to v4. To resolve this, re-process your simulation in Rocky v4 or later.

PARTICLE AND DIVISIONS TAGGING PARAMETER DEFINITIONS

See the images and tables below for more information on Particle Tagging, including Divisions Tagging.

To access the Particles tagging option, right-click the Particles option in the Data Panel and then click Particles Calculations, Tagging and Particles, or access the Particles Calculations option in Calculations. As illustrated in the images below:

Figure 5.190: Particle Tagging option on the Data panel inside Calculations



Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

Figure 5.191: Particle Tagging option on the Data panel inside Particles

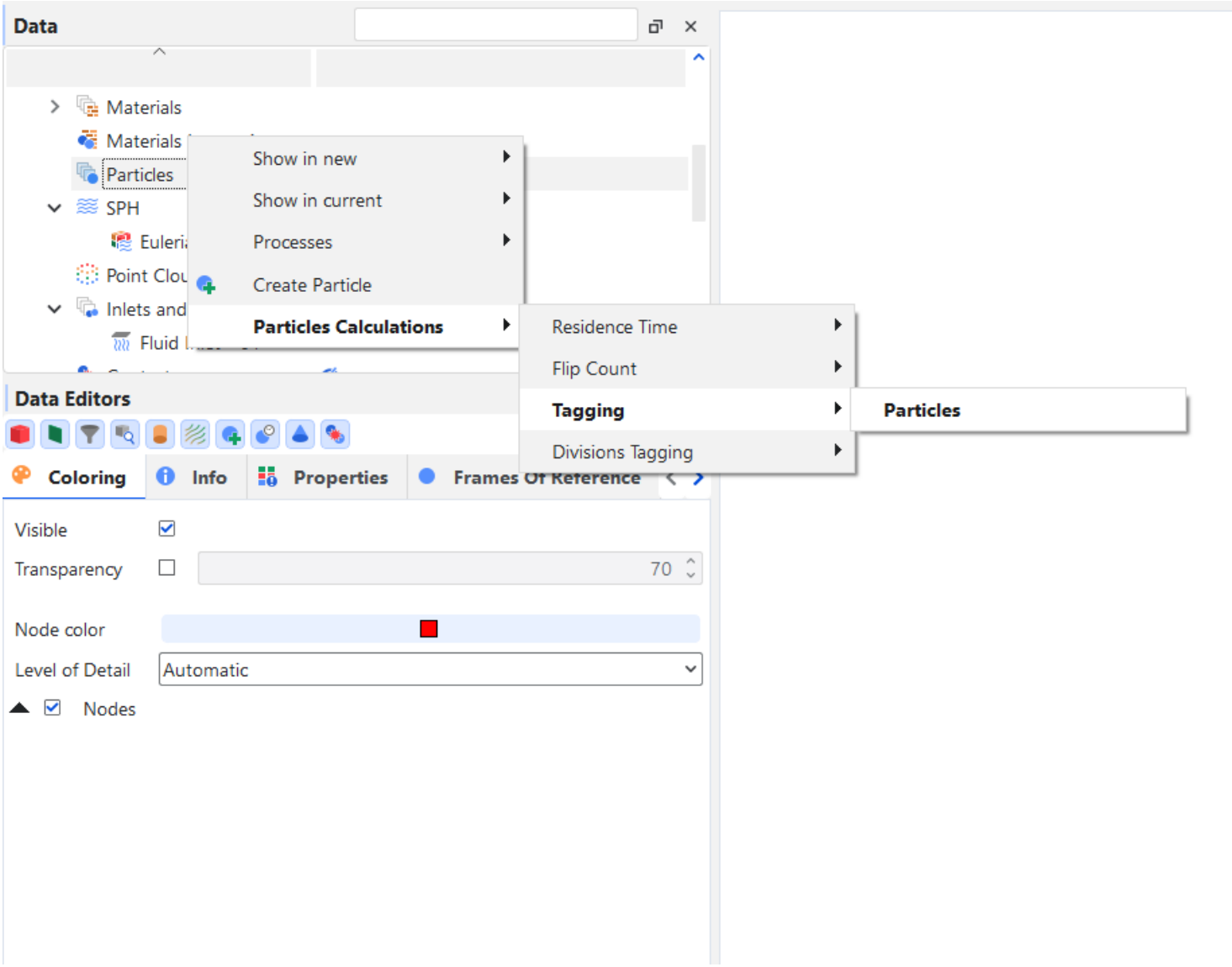
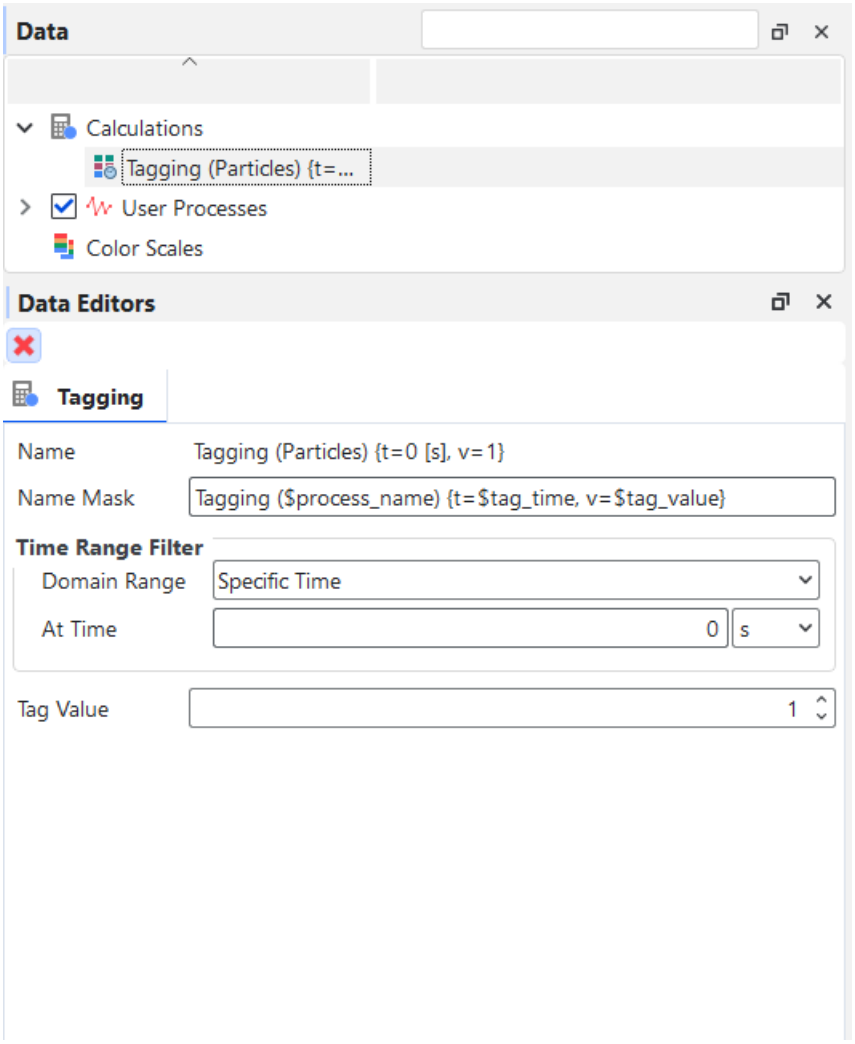


Figure 5.192: Particle Tagging on the Data Editors panel



Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

Figure 5.193: Particle Calculations, Divisions Tagging options (for a Cube) on the Data Editors panel

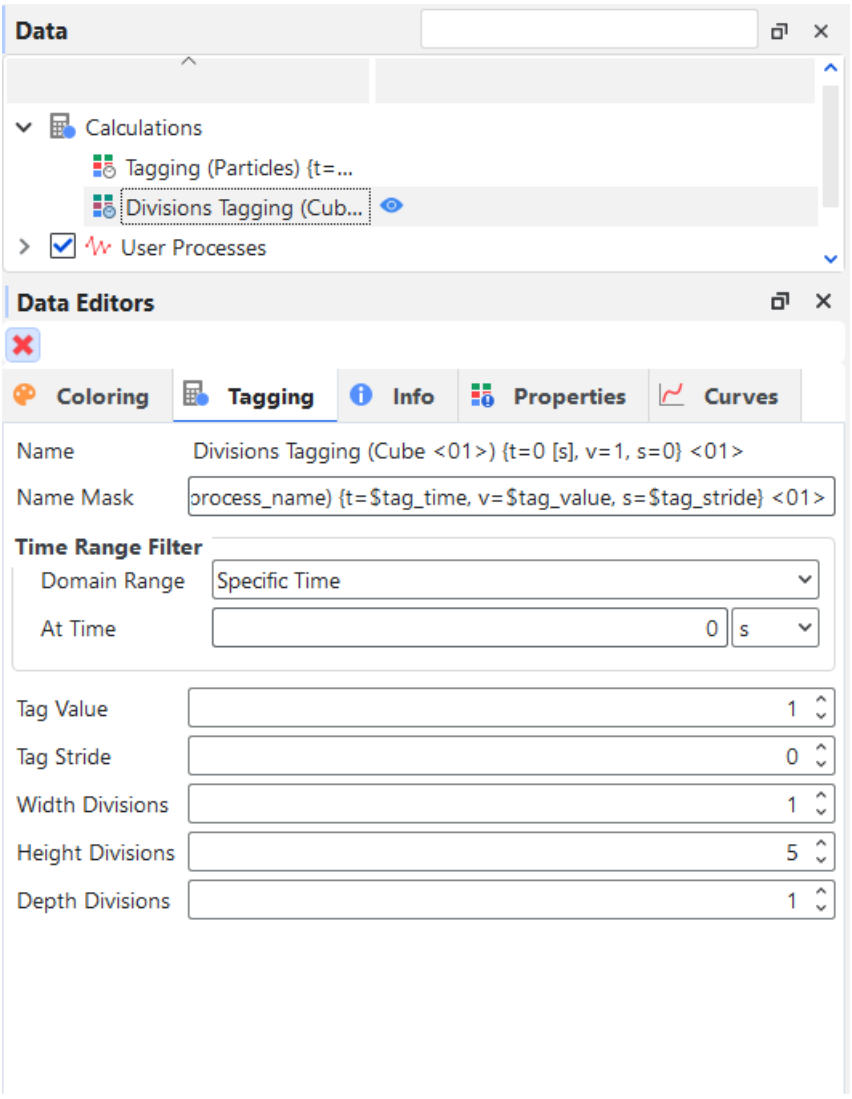


Table 1: Particle Calculations, Tagging and Divisions Tagging options

Setting	Description	Range
Name	The unique identifier that is displayed in the Properties list and also on the Color Bar when the Property is dragged to a 3D View. By using the **Name Mask** setting, you may change the Name to whatever you want but by default, the Name is provided in the format below: • For Tagging: Tagging($process_name) {t=$tag_time, v=$tag_value}	Display only

Setting	Description	Range
	• For Divisions Tagging: Divisions Tagging($process_name) {t=$tag_time, v=$tag_value, s=$tag_stride} **Tip:** See **Name Mask** below for Keyword definitions.	
Name Mask	Method by which you can change the **Name** using text that you type or Keywords that you enter. Any Keywords you use will be replaced with the corresponding data in the **Name** field, and will be updated when the data supporting the Keyword changes. The Keywords available include the following: • **$process_name**: The name of the User Process to which Tagging was applied. • **$tag_time**: A description of the Time Range Filter you have applied to the Tagging calculation. • **$tag_value**: The **Tag Value** you have assigned to the tag. • **$tag_stride**: The **Tag Stride** value. **Note:** This is only for Divisions Tagging.	No limit
Time Range Filter		
Domain Range	Defines what time steps of the simulation are included in the tagging selection. The options are as follows: • **Application Time Filter**: Use the range settings on the Time Filter dialog to limit the time steps used. (For more information, see About the Time Toolbar (p. 1279)). • **All**: Use all available time steps in the simulation. • **Last Output**: Use only the very last (most recent) time step that was calculated. • **Time Range**: Limit the time step values by a specific range that you choose. (Uses the **Initial** and **Final** values you set.) • **Specific Time**: Choose only a single specific time step. (Uses the **At Time** value you set.)	Application Time Filter; All; Last Output; Time Range; Specify Time; After Time; Time Range Relative to Simulation End

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

Setting	Description	Range
	• **After Time**: Use all time steps available after a specific time period of your choosing. This selection will include any particles that will enter the User Process at any point after the **Initial** value you set. • **Time Range Relative to Simulation End**: Specify how long before the very end of the simulation to include time steps. (Uses the **Range from end** value you set.)	
Initial	When **Time Range** or **After Time** is chosen for **Domain Range**, this is the starting time to begin tagging within the selection.	Any value between 0 and the final simulation time.
Final	When **Time Range** is chosen for **Domain Range**, this is the ending time when tagging within the selection is stopped.	Any value between the **Initial** time and the final simulation time.
At Time	When **Specific Time** is chosen for **Domain Range**, this is the exact moment in which the Particles within the selection will be tagged.	Any value between 0 and the final simulation time.
Range from end	When **Time Range Relative to Simulation End** is chosen for **Domain Range**, this is the period of time before the final simulation time in which time steps will be included in the tagging selection. For example, when you want to include only the last X seconds of the simulation.	Any value between 0 and the final simulation time.
Tag Value	The number assigned to every "tagged" particle within the selection and time range specified. For **Tagging**, this will be a single number. For **Divisions Tagging**, there can be one unique number per block. **Note:** All other particles not within the tagging selection will retain the value of 0 (zero).	Whole positive numbers
Divisions Tagging Only		
Tag Stride	Enables you to assign **Tag Values** for your blocks in a striation rather than sequentially. The stride begins with the value you set for **Tag Value** and assigns the striated values thereafter. Example: Assuming a **Tag Value** of 1:	Whole positive numbers

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

Setting	Description	Range
	• A value of 0 will have no stride and will result in the blocks being numbered sequentially. *1, 2, 3, 4, 5…* Note: This is the default value. • A value of 1 will result in all blocks having the same number. *1, 1, 1, 1, 1…* • A value of 2 will result in the blocks being numbered by 1s and 2s. *1, 2, 1, 2, 1…* • A value of 3 will result in the blocks being numbered by 1s through 3s. *1, 2, 3, 1, 2, 3, 1…* Example: Assuming a **Tag Value** of 4: • A value of 2 will result in the blocks being numbered by 4s and 5s. *4, 5, 4, 5, 4…* • A value of 3 will result in the blocks being numbered by 4s through 6s. *4, 5, 6, 4, 5, 6, 4…*	
Width Divisions	When based upon a Cube User Process, this is the amount of blocks you want the Cube divided into in the X direction (local; corresponds to the red directional handle on the Cube).	Whole values greater than or equal to 1
Height Divisions	When based upon a Cube User Process, this is the amount of blocks you want the Cube divided into in the Y direction (local; corresponds to the green directional handle on the Cube).	Whole values greater than or equal to 1
Depth Divisions	When based upon a Cube User Process, this is the amount of blocks you want the Cube divided into in the Z direction (local; corresponds to the blue directional handle on the Cube).	Whole values greater than or equal to 1

SPH TAGGING

Tagging marks with a number a given portion of SPH elements in a given time and can be used to show attributes such as all SPH elements inside a cube in time t = 1s with number 1, or all SPH elements that have temperature above 300K on time t = 10s and many other cases following the same logic.

To access the SPH tagging option, right-click the SPH option in the Data Panel and then click SPH Calculation, Tagging and chose the entity you want to apply the tagging - either a user process such as a cube, the SPH entity itself, or access the SPH Calculations option in Calculations. As illustrated in the images below:

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

Figure 5.194: SPH Tagging option on the Data panel inside Calculations

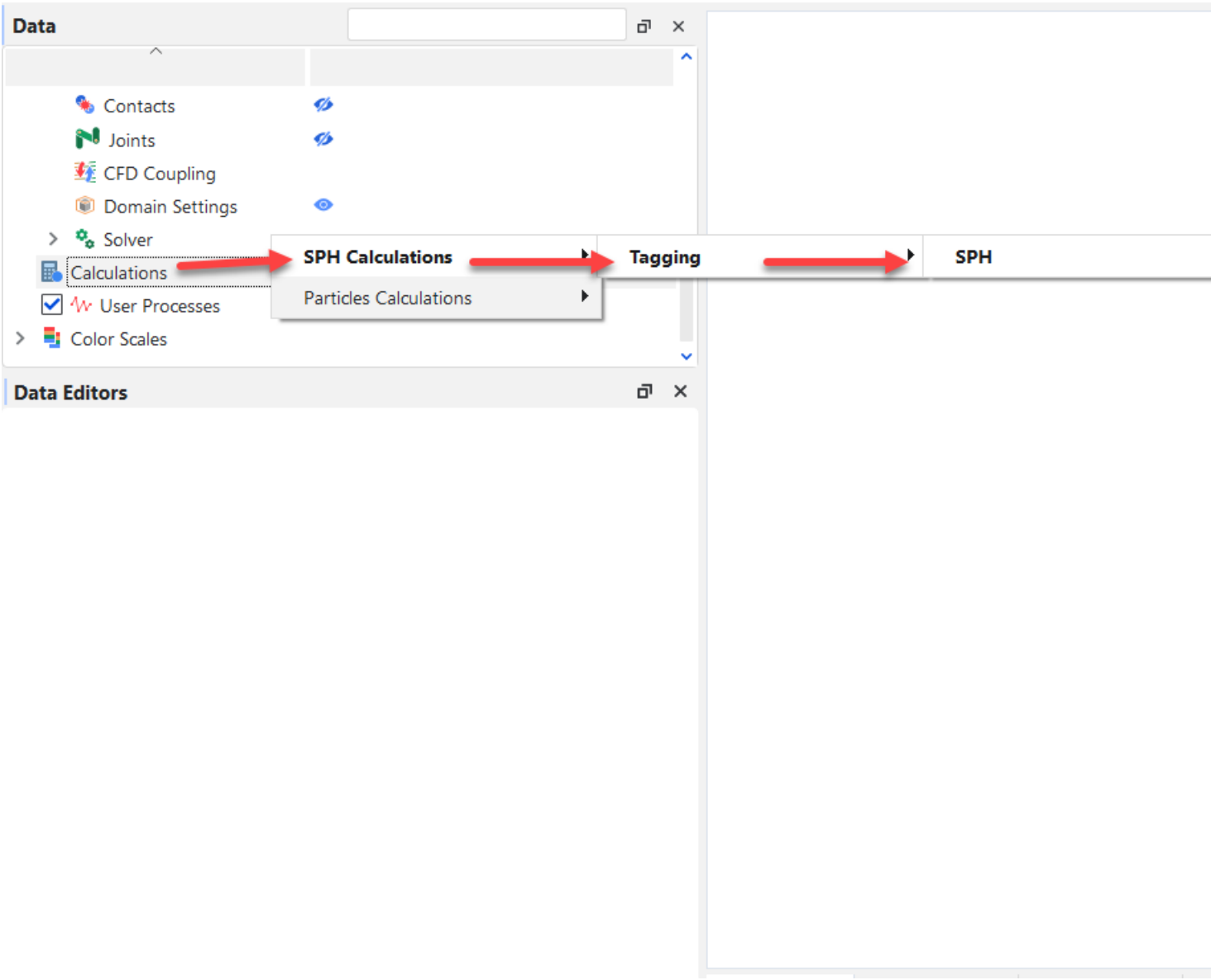
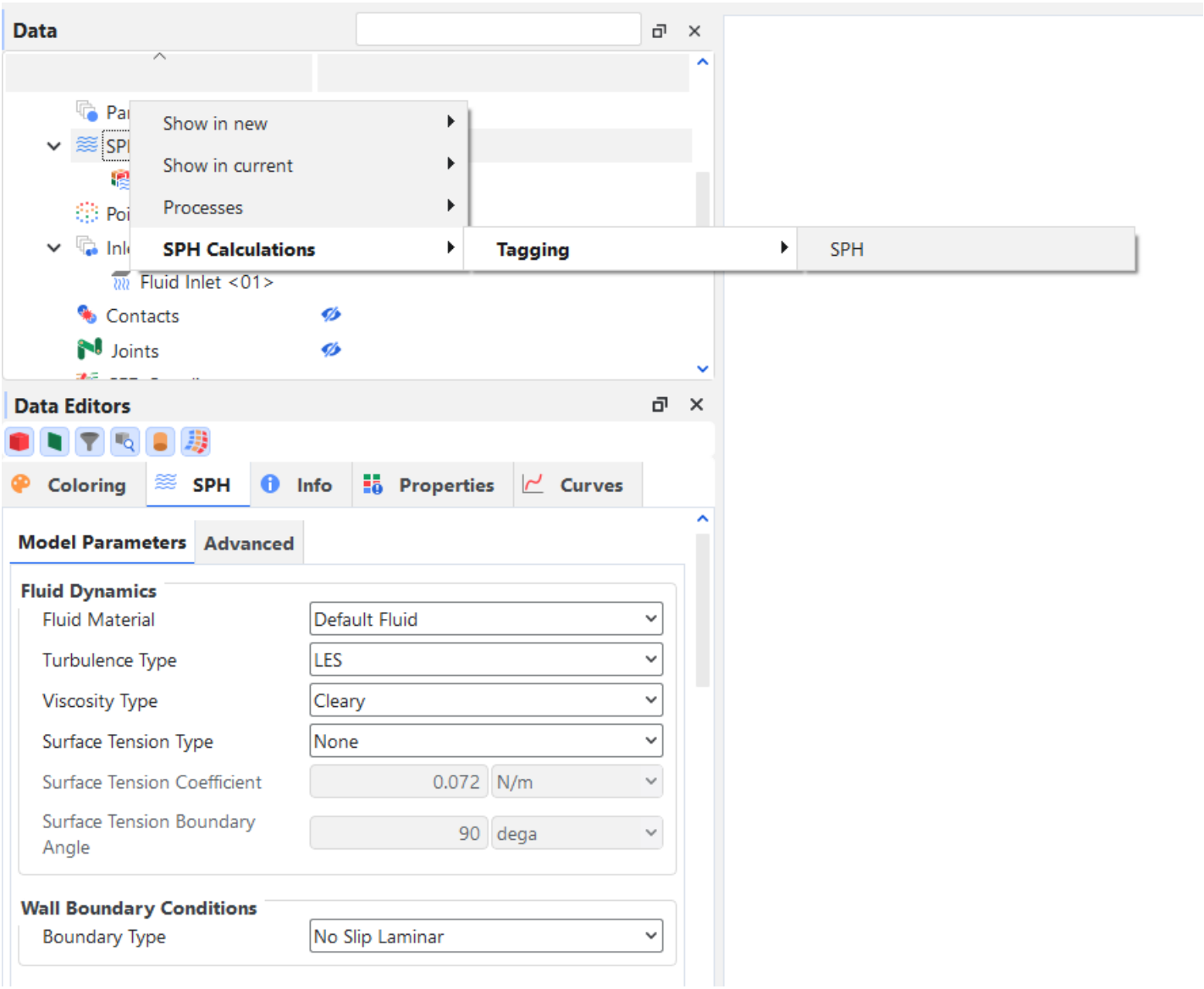


Figure 5.195: SPH Tagging option on the Data panel inside SPH



Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

Figure 5.196: SPH Tagging on the Data Editors panel

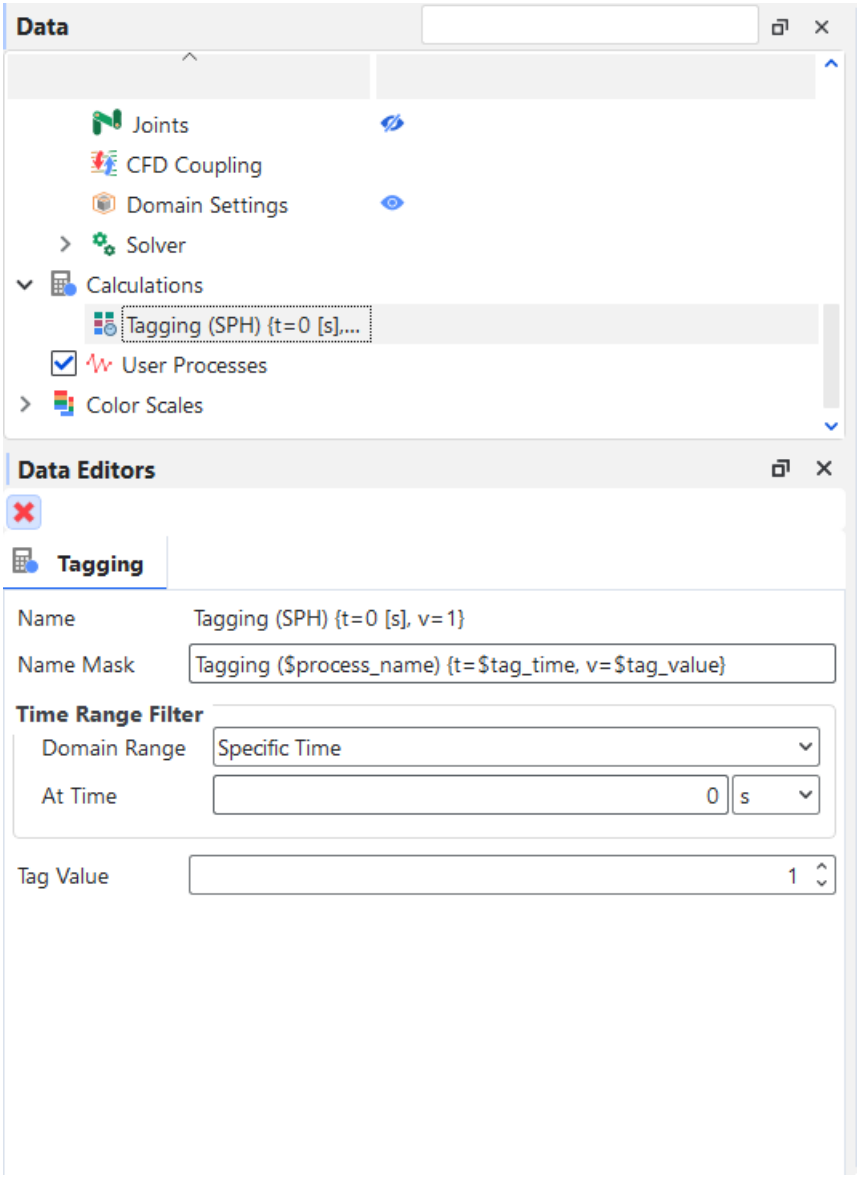


Table 2: Calculations, SPH Tagging options

Setting	Description	Range
Name	The unique identifier that is displayed in the Properties list and also on the Color Bar when the Property is dragged to a 3D View. By using the **Name Mask** setting, you may change the Name to whatever you want but by default, the Name is provided in the format below • SPH Tagging: Tagging(SPH) {t=0 [s], v=1}	Display only
Name Mask	Method by which you can change the **Name** using text that you type or Keywords that you enter. Any Keywords you	No limit

Setting	Description	Range
	use will be replaced with the corresponding data in the **Name** field, and will be updated when the data supporting the Keyword changes. The Keywords available include the following: • **$process_name**: The name of the User Process to which Tagging was applied. • **$tag_time**: A description of the Time Range Filter you have applied to the Tagging calculation. • **$tag_value**: The **Tag Value** you have assigned to the tag.	
Time Range Filter		
Domain Range	Defines what time steps of the simulation are included in the tagging selection. The options are as follows: • **Application Time Filter**: Use the range settings on the Time Filter dialog to limit the time steps used. (For more information, see About the Time Toolbar (p. 1279)). • **All**: Use all available time steps in the simulation. • **Last Output**: Use only the very last (most recent) time step that was calculated. • **Time Range**: Limit the time step values by a specific range that you choose. (Uses the **Initial** and **Final** values you set.) • **Specific Time**: Choose only a single specific time step. (Uses the **At Time** value you set.) • **After Time**: Use all time steps available after a specific time period of your choosing. This selection will include any particles that will enter the User Process at any point after the **Initial** value you set. • **Time Range Relative to Simulation End**: Specify how long before the very end of the simulation to include time steps. (Uses the **Range from end** value you set.)	Application Time Filter; All; Last Output; Time Range; Specify Time; After Time; Time Range Relative to Simulation End.
Initial	When **Time Range** or **After Time** is chosen for **Domain Range**, this is the starting time to begin tagging within the selection.	Any value between 0 and the final simulation time.

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

Setting	Description	Range
Final	When **Time Range** is chosen for **Domain Range**, this is the ending time when tagging within the selection is stopped.	Any value between the **Initial** time and the final simulation time.
At Time	When **Specific Time** is chosen for **Domain Range**, this is the exact moment in which the Particles within the selection will be tagged.	Any value between 0 and the final simulation time.
Range from end	When **Time Range Relative to Simulation End** is chosen for **Domain Range**, this is the period of time before the final simulation time in which time steps will be included in the tagging selection. For example, when you want to include only the last X seconds of the simulation.	Any value between 0 and the final simulation time.
Tag Value	The number assigned to every "tagged" particle within the selection and time range specified. For **Tagging**, this will be a single number.	Whole positive numbers

What would you like to do?

- Apply Tagging or Divisions Tagging to a Particles User Process Selection (p. 1270)

See Also:

- Filter Views and Data with User Processes (p. 1174)

Apply Particles Calculations to a Particles User Process Selection

Note:

This procedure is applicable to the following types of Particle Calculations: Flip Count and Residence Time. To apply Tagging or Divisions Tagging, please see Apply Tagging or Divisions Tagging to a Particles User Process Selection (p. 1270).

1. Ensure that you have created a Cube, Cylinder, Polyhedron, Plane, or Properties User Process from the Particles entity (p. 1395), and have sized and positioned it the way you want. (See also Filter Views and Data with User Processes (p. 1174).)

2. From the **Data** panel, right-click **Particles Calculations**, point to **Calculations**, point to the type of calculation you want, and then from the sub-menu that appears, click the name of the User Process to which you want the calculation applied. A new item appears under **Particles Calculations** in the **Data** panel. New calculation properties appended with the User Process name appear in the Properties lists for both the User Process and the Particles entity.

Tips:

- If in step 2 above, no User Processes appears in the sub-menu for the calculation you chose, it is likely that step 1 has not been completed correctly. (See also About Particles Calculations (p. 1242).)

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

- To visualize the new Property, click and drag it to a 3D View window. (See also Create and Modify a 3D View (p. 1040).)
- If the particles aren't appearing in the 3D View the way you expect, ensure that the User Process and Particles entities are shown or hidden as you want. (See also Show/Hide Components by Using Eye icons and Checkboxes (p. 113).)

See Also:

- About Particles Calculations (p. 1242)
- About Particle Tagging (p. 1248)
- Filter Views and Data with User Processes (p. 1174)

Apply Tagging or Divisions Tagging to a Particles User Process Selection

Notes:

- This procedure is applicable to the following types of Particle Calculations: Tagging and Divisions Tagging. To apply other types of Particle Calculations, please see Apply Particles Calculations to a Particles User Process Selection (p. 1269).
- Neither Tagging nor Divisions Tagging are compatible with Particle Breakage Calculations processed with versions prior to Rocky v4. To resolve this issue, re-process your simulation with Rocky v4 or later. (See also Enable Particle Breakage Calculations (p. 397).)

1. Do one of the following:
   - For Tagging calculations, ensure that you have done one of the following:
     - Selected the Particles entity.
     - Created a Cube, Cylinder, Polyhedron, Plane, or Properties User Process from the Particles entity (p. 1395), and have sized and positioned it the way you want. (See also Filter Views and Data with User Processes (p. 1174).)
   - For Divisions Tagging calculations, ensure that you have created either a Cube or Cylinder User Process from the Particles entity (p. 1395), and have sized and positioned it the way you want. (See also Analyze Data within a Defined Shape (p. 1175).)
2. From the **Data** panel, right-click **Particles Calculations**, point to **Calculations**, point to either **Tagging** or **Divisions Tagging**, and then from the sub-menu that appears, click the name of the User Process to which you want the calculation applied. A new item appears under **Particles Calculations** in the **Data** panel.
3. Under **Particle Calculations** in the **Data** panel, select the new entry you just added.
4. From the **Data Editors** panel, select the **Tagging** tab and then set the options as you want. New properties named with the **Name** value appear in the Properties lists for the User Process shape you selected, the Particles entity, and any other User Process you create from the Particles entity. Tip:To change the name displayed for your Tagging calculation, use the **Name Mask** field.

Tips:

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

- If in step 2 above, no User Processes appears in the sub-menu for the Tagging calculation you chose, it is likely that step 1 has not been completed correctly. (See also About Particle Tagging (p. 1248).)
- To visualize the new Property, click and drag it to a 3D View window. (See also Create and Modify a 3D View (p. 1040).)
- If the tagged particles aren't appearing in the 3D View the way you expect, ensure that the User Process and Particles entities are shown or hidden as you want. (See also Show/Hide Components by Using Eye icons and Checkboxes (p. 113).)
- To have particle fragments (p. 1393) inherit the tag value of their parent particle, tag particles prior to them breaking; to have fragments treated individually, tag particles after they have broken.

See Also:

- About Particle Tagging (p. 1248)
- About Particles Calculations (p. 1242)
- Filter Views and Data with User Processes (p. 1174)

5.5.14. Analyze Contacts from a Sub-Set of Particles

Important:

Even though both the **Contact to Particle** and the **Particle to Contact** User Processes will always be available, there will be no Contacts data to analyze unless you turn on the collection of Contacts data *prior to* processing your simulation. (See also Turn On Contacts Data Collection (p. 756).)

The **Particle to Contact** User Process is designed to filter out which contacts are happening to a particular sub-set of particles. Therefore, this User Process is created from the main **Particles** entity (p. 1395) or from another User Process based upon Particles.

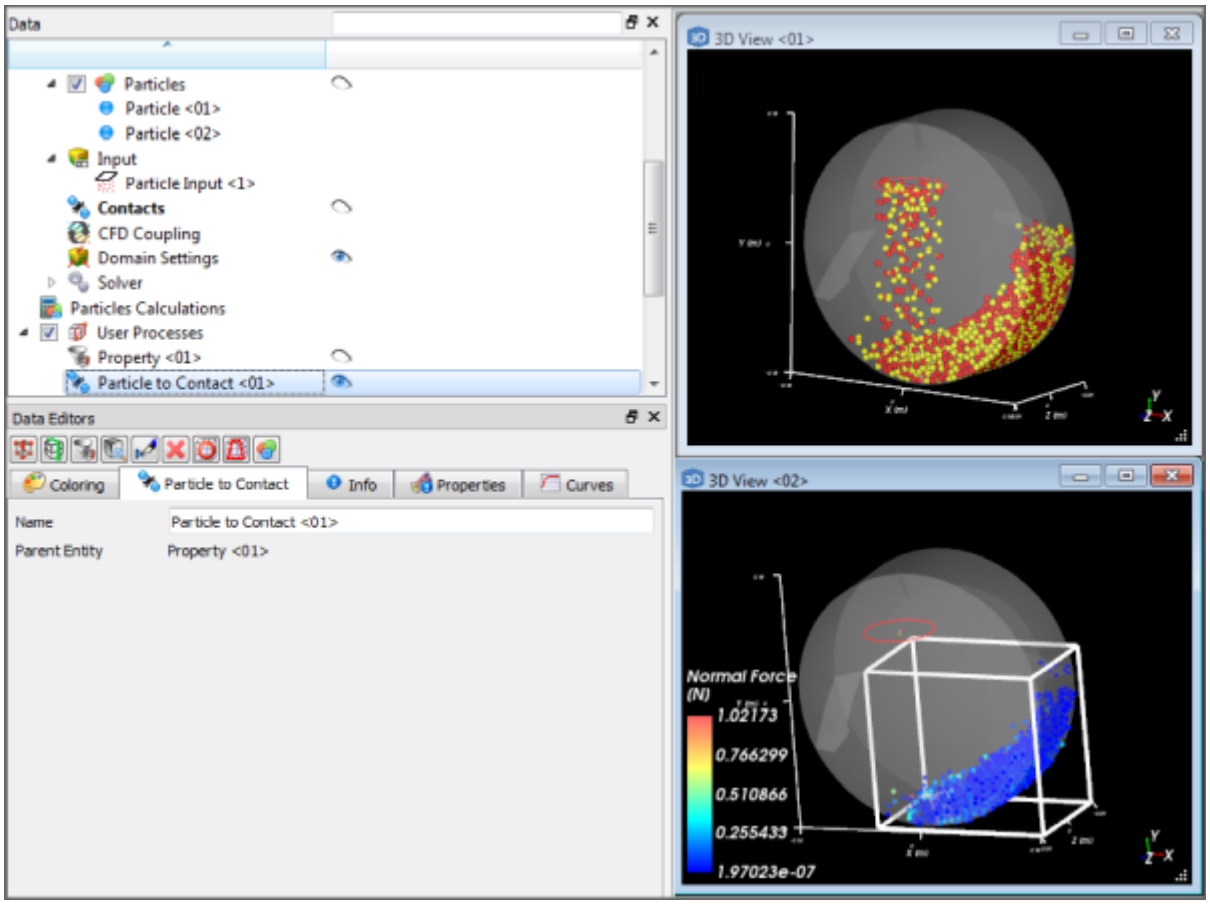
Even though you create this User Process from Particles, its linked entity is **Contacts**. This means that after applying this User Process, you are able to further analyze the results by Contacts Properties. (See also About Properties (p. 933).)

Having a different linked entity than the entity you originally created it from can make determining its relationship to other entities challenging. So for this User Process (and its sibling, **Contact to Particle**), the item in **bold** is its "linked" entity in the sense that it indicates which Properties and functions the User Process adopts. But the entity it was created from (in most cases, referred to as the "parent") is listed in the **Data Editors** panel on the **Particle to Contact** tab.

For example, in the image below, **Property <01>** was created from the main **Particles** entity, and then **Particle to Contact <01>** was created from **Property <01>**. This originator-type relationship is indicated on the **Particle to Contact** tab next to **Parent Entity**.

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

Figure 5.197: Property process created from Particles (top view) showing desired particle group in yellow; Particle to Contact process created from Property process showing Normal Forces (bottom view)



Contacts is listed in bold on the **Data** panel because it indicates which Properties and functions the User Process adopts.

Usage Examples

There are several ways in which you can use this User Process. For example, you can:

- Filter particles of a particular particle group and then use the Particle to Contact User Process to analyze normal forces (Figure 5.197: Property process created from Particles (top view) showing desired particle group in yellow; Particle to Contact process created from Property process showing Normal Forces (bottom view) (p. 1272)) or overlaps from collisions involving the desired particle shape.
- Filter particles with velocity in a given range and then use the Particle to Contact User Process to analyze normal forces or overlaps from collisions involving them.
- Filter particles with given size range and then use the Particle to Contact User Process to get the histogram of normal and tangential forces or overlaps from collisions involving them.

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

Particle to Contact Parameters

See the table below and the images above to understand the parameters for the Particle to Contact User Process.

Table 1: Particle to Contact Parameter Definitions

Setting	Description	Range
Name	Enables you to specify a unique identifier for the User Process. **Note:** This same name will appear in the **Data** panel under **User Processes**.	No limit
Parent Entity	Lists the name of the entity from which the selected User Process was created. Can be either the main Particles entity or a User Process based upon Particles. **Note:** Unlike with other User Processes, the entity from which this User Process originated does not share its Properties and functions with the User Process. Rather, the item listed in **bold** on **Data** panel is the entity for which Properties and are based.	Display only

What would you like to do?

- Learn more About Contacts (p. 746)
- Create a Particle to Contact User Process (p. 1273)

Create a Particle to Contact User Process

Important:

Even though both the **Contact to Particle** and the **Particle to Contact** User Processes will always be available, there will be no Contacts data to analyze unless you turn on the collection of Contacts data *prior to* processing your simulation. (See also Turn On Contacts Data Collection (p. 756).)

1. From the **Data** panel, right-click the Particles-based entity (p. 1395) upon which you want to base the new User Process, point to **Processes**, and then click **Particle to Contact**.
2. From the **Data Editors** panel, on the **Particle to Contact** tab, enter or view the options you want.

Tips:

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

- Use the **Properties** tab to analyze Contacts Properties for this sub-set of contacts. (See also About Properties (p. 933).)
- Use the **Colorings** tab to change colors and visualization settings for this sub-set of contacts. (See also About the Coloring Tab (p. 93).)
- Use the **Data** panel eye icons to hide entities from the 3D View that may be covering the display of this User Process. (See also Show/Hide Components by Using Eye icons and Checkboxes (p. 113).)

See Also:

- Turn On Contacts Data Collection (p. 756).
- About Contacts (p. 746)
- Analyze Contacts from a Sub-Set of Particles (p. 1271)
- Filter Views and Data with User Processes (p. 1174)

5.5.15. Analyze Particles from a Sub-Set of Contacts

Important:

Even though both the **Contact to Particle** and the **Particle to Contact** User Processes will always be available, there will be no Contacts data to analyze unless you turn on the collection of Contacts data *prior to* processing your simulation. (See also Turn On Contacts Data Collection (p. 756).)

The **Contact to Particle** User Process is designed to filter out which particles are experiencing a particular sub-set of contacts, and therefore, is created from the main **Contacts** entity (p. 1395) or from another User Process based upon Contacts.

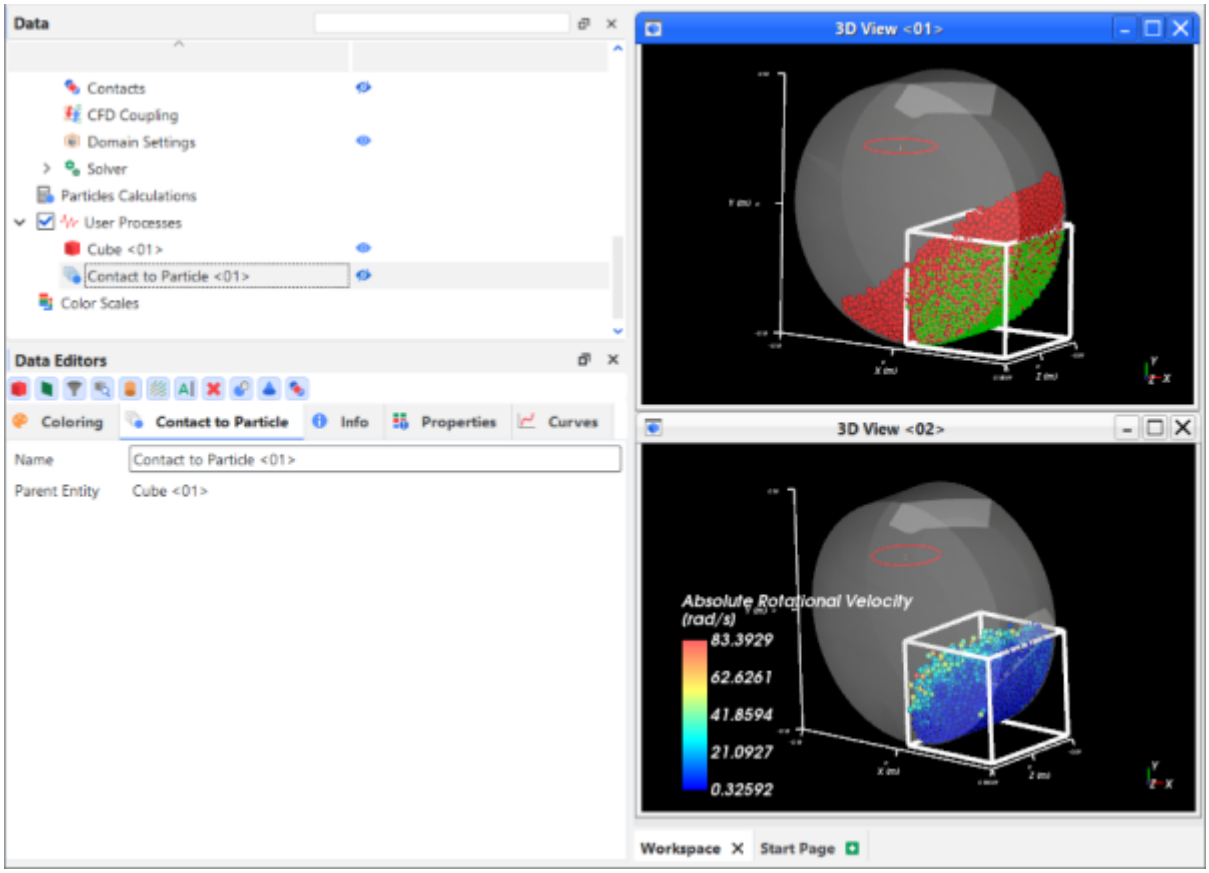
Even though you create this User Process from Contacts, its linked entity is **Particles**. This means that after applying this User Process, you are able to further analyze the results by Particles Properties. (See also About Properties (p. 933).)

Having a different linked entity than the entity you originally created it from can make determining its relationship to other entities challenging. So for this User Process (and its sibling, **Particle to Contact**), the item in **bold** is its "linked" entity in the sense that it indicates which Properties and functions the User Process adopts. But the entity it was created from (in most cases, referred to as the "parent") is listed in the **Data Editors** panel on the **Contact to Particle** tab.

For example, in the image below, **Cube <01>** was created from the **Contacts** entity, and then **Contact to Particle <01>** was created from **Cube <01>**. This originator-type relationship is indicated on the **Contact to Particle** tab next to **Parent Entity**.

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

Figure 5.198: Cube from Contacts (in green, top view); Contact to Particle from Cube colored by Absolute Rotational Velocity (bottom view)



The main **Particles** entity is listed in bold on the **Data** panel because it indicates which Properties and functions the User Process adopts.

Usage Examples

There are several ways in which you can use this User Process. For example, you can:

- **Analyze the contacts happening on a specific region of your domain.** To do this, create a cube process on contacts and then use the Contact to Particle User Process to investigate the velocity of the particles (Figure 5.198: Cube from Contacts (in green, top view); Contact to Particle from Cube colored by Absolute Rotational Velocity (bottom view) (p. 1275)), or the particle size distribution.
- **Analyze large overlaps occurring in a certain region.** To do this, create a property process upon contacts and then filter only contacts with overlap greater than a certain value. Then, create a Contact to Particle process to investigate the shape of the particles that are involved in these high overlap collisions.
- **Investigate which particles are involved on collisions with large, wear-causing tangential forces.** To do this, create a property process upon contacts and then filter only the contacts with large forces. Then, create a Contacts to Particle process in order to get the details of the particles involved—for example, the group, size, or even starting location (when combined with the tagging process).

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

Contact to Particle Parameters

See the table below and the image above to understand the parameters for the Contact to Particle User Process.

Table 1: Contact to Particle Parameter Definitions

Setting	Description	Range
Name	Enables you to specify a unique identifier for the User Process. **Note:** This same name will appear in the **Data** panel under **User Processes**.	No limit
Parent Entity	Lists the name of the entity from which the selected User Process was created. Can be either the Contacts entity or a User Process based upon Contacts. **Note:** Unlike with other User Processes, the entity from which this User Process originated does not share its Properties and functions with the User Process. Rather, the item listed in **bold** on **Data** panel is the entity for which Properties and are based.	Display only

What would you like to do?

- Learn more About Contacts (p. 746)
- Turn On Contacts Data Collection (p. 756)
- Create a Contact to Particle User Process (p. 1276)

Create a Contact to Particle User Process

Important:

Even though both the **Contact to Particle** and the **Particle to Contact** User Processes will always be available, there will be no Contacts data to analyze unless you turn on the collection of Contacts data *prior to* processing your simulation. (See also Turn On Contacts Data Collection (p. 756).)

1. From the **Data** panel, right-click the Contacts-based entity (p. 1395) upon which you want to base the new User Process, point to **Processes**, and then click **Contact to Particles**.

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

2. From the **Data Editors** panel, on the **Contact to Particles** tab, enter or view the options you want.

Tips:

- Use the **Properties** tab to analyze Particle Properties for this sub-set of particles. (See also About Properties (p. 933).)
- Use the **Colorings** tab to change colors and visualization settings for this sub-set of particles. (See also About the Coloring Tab (p. 93).)
- Use the **Data** panel eye icons to hide entities from the 3D View that may be covering the display of this User Process. (See also Show/Hide Components by Using Eye icons and Checkboxes (p. 113).)

See Also:

- Turn On Contacts Data Collection (p. 756).
- About Contacts (p. 746)
- Analyze Particles from a Sub-Set of Contacts (p. 1274)
- Filter Views and Data with User Processes (p. 1174)

5.5.16. Surface User Process

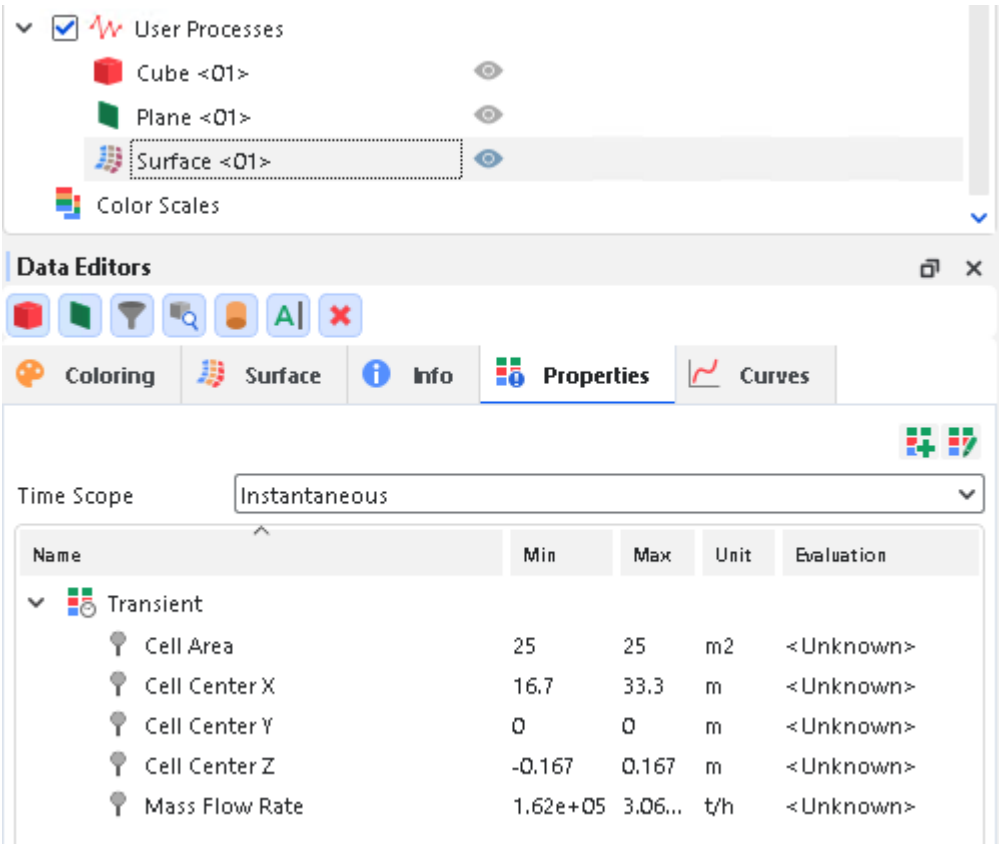
The Surface User Process enables you to import a custom .stl surface which after can be used to measure SPH mass flow rate in post-processing phase.

Its function is similar to SPH Mass Flow Rate Module (p. 887), but it's important to remember that User Process is based upon calculations that were already done, making it possible to work with the relevant data collected after the simulation.

Use the figure below to see some information provided by Surface User Process:

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

Figure 5.199: Surface User Process Properties



See Also:

- Filter Views and Data with User Processes (p. 1174)

5.6. Change the Current Output by Using the Time Toolbar

In Rocky, the output time (p. 1393) refers to any point in the simulation when an output file (p. 1393) has been saved. By selecting different output times or by playing them in succession, you can analyze values calculated at different times within your simulation.

What would you like to do?

- Learn more About the Time Toolbar (p. 1279)
- Use the Time Toolbar to Change the Current Output (p. 1280)
- Limit Output Values by a Specific Time Range (p. 1281)

See Also:

- Analyzing a Simulation (p. 931)

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

5.6.1. About the Time Toolbar

The Time toolbar (shown below) is the where you change what simulation output time (p. 1393) (also referred to as time step (p. 1395)) is currently being analyzed. What output time you select changes the values displayed in 3D Views, individual statistics, and animations, and determines the outcomes of settings like Particle Trajectories (p. 1209).

Figure 5.200: Time toolbar showing the output time at 0.06s (3rd Output file)



The Time toolbar enables you to select an individual output time based on the simulation time or output file, or play all available outputs sequentially in a loop. You can also limit the outputs shown on the Time toolbar to a certain range that you specify by using the Time Filter. This can be useful when using the Particles Time Selection User Process (p. 1221), defining Output variables (p. 250), or when tagging Particles (p. 1248).

Note:

Unless you have enabled the **Auto Refresh** option, the ouput values are updated during processing only when you choose to manually Refresh the results. (See also About Starting a Simulation (p. 908).)

Previewing Motions Using the Time Toolbar

If you haven't yet processed your simulation, the slider on the Time toolbar will be highlighted yellow (as shown below) to indicate that it can be used for the Motion Preview window. (See also About Creating and Applying Motion Frames to Imported Geometries (p. 533).)

Figure 5.201: Time Toolbar with slider highlighted in yellow to indicate its use during Motion Preview



When the slider is highlighted yellow, the toolbar represents only the motion-related preview of the expected output times, and does not reflect any other particle calculations that have been performed. A preview length equal to the **Simulation Duration** is provided (see also About Solver Parameters (p. 464)) with an upper limit of no more than 30,000 time steps (p. 1395).

After processing, the Time toolbar goes back to its original function, and the highlighting is removed.

Use the images and table below to help you understand how to use the Time toolbar and Time filter to change the current time step.

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

Figure 5.202: Time Filter dialog

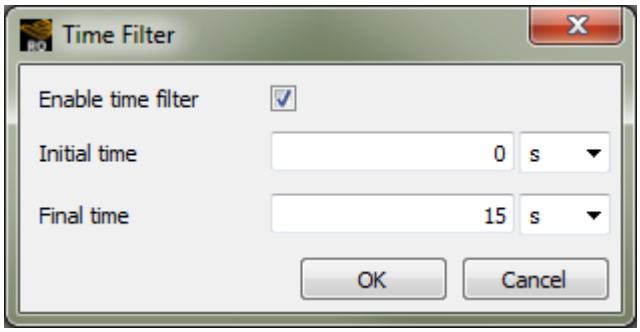


Table 1: Time Filter dialog options

Setting	Description	Range
Enable time filter	When selected, allows you to define a time range by which to limit what outputs are displayed on the Time toolbar.	Turns on or off
Initial time	Enables you to set the beginning value for the time range.	Any value
Final time	Enables you to set the ending value for the time range.	Any value

What would you like to do?

- Use the Time Toolbar to Change the Current Output (p. 1280)
- Limit Output Values by a Specific Time Range (p. 1281)
- Learn more output time (p. 1393) you want to display.

5.6.2. Use the Time Toolbar to Change the Current Output

From the **Time** toolbar, use the **Time** list, slider, or other navigation buttons to select the output time (p. 1393) you want to display.

Note:

If you aren't seeing as many outputs as you expected, you might need to refresh your simulation results (see also About Starting a Simulation (p. 908)) or disable your Time Range (see also Limit Output Values by a Specific Time Range (p. 1281).)

Tip:

To play a preview of your simulation, select a 3D View and then click the **Play simulation** button.

See Also:

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

- About the Time Toolbar (p. 1279)

5.6.3. Limit Output Values by a Specific Time Range

1. From the **Time** toolbar, click the **Timeset filter** button.
2. From the **Time Filter** dialog, do all of the following:
   a. Enable the **Enable time filter** check box.
   b. Define the time range by entering the values you want in the **Initial time** and **Final time** boxes.
   c. Click **OK**.

The output times (p. 1393) available for display on the **Time** toolbar are now limited by the range you defined.

Tip:

To remove the filter and show all available output times, from the **Time Filter** dialog, clear the **Enable time filter** checkbox, and then click **OK**.

See Also:

- About the Time Toolbar (p. 1279)
- Use the Time Toolbar to Change the Current Output (p. 1280)

5.6.4. Monitor Analyses Statuses by Using the Progress Panel

Rocky is a complex program that enables you to perform various types of analyses on your simulation data. The more complex the analysis request and the more requests you make at once, the longer it will take for Rocky to calculate and display them.

To help you keep track of these more complex or time-consuming requests, Rocky includes a Progress panel, which displays the status of the active processes you have requested.

What would you like to do?

- Learn more About the Progress Panel (p. 1281)
- Learn more About Panels (p. 78)
- Stop an Active Process (p. 1283)

5.6.5. About the Progress Panel

The Progress panel enables you to monitor the status of your active requests for more complex calculations and/or display changes. Computationally intensive requests like generating an animation, or creating several graphs at once, can take additional time to process. The Progress panel enables you to see the status of these requests while still working on other tasks.

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

While the information on the Progress panel is display only, you can use the information provided to decide whether or not to take action on a task from the original request location. For example, if you notice that your animation is taking a long time to generate and you want to delay that task until a later point, you may cancel generating the animation by finding the animation window and closing it. The same is true for other request windows; you can stop the request by closing them.

Notes:

- If the request you want to close is dependent upon another request, you will be unable to close it until the dependency is cleared.
- Because there are already several ways for you to monitor the progress of your simulation (see also About Starting a Simulation (p. 908)), the Progress panel does not display this information.

Use the image and table below to help you understand how you can use the Progress panel to monitor the statuses of your analyses requests.

Figure 5.203: Progress panel showing the status of a Time plot request

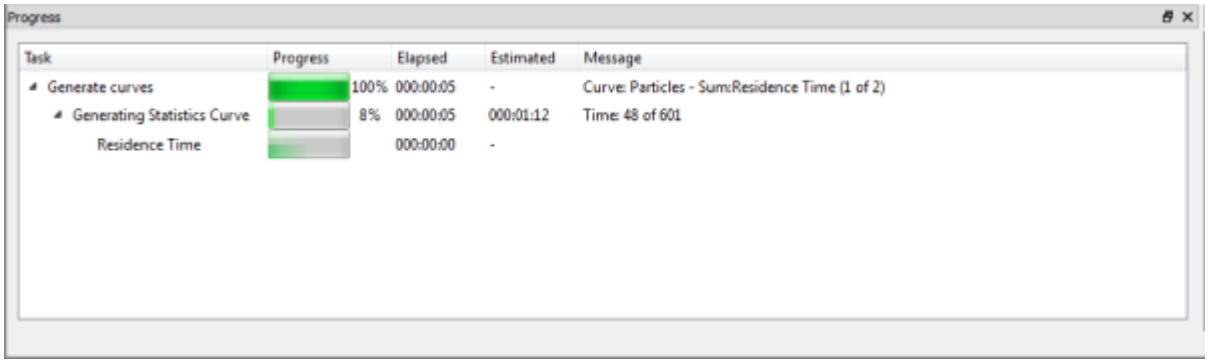


Table 1: Progress panel options

Column Title	Description
Task	Name given by Rocky to the tasks and sub-tasks required to complete the request.
Progress	Indicates both visually and by percentage how much of the task is complete.
Elapsed	The amount of real time in hours minutes, and seconds since the task started processing.
Estimated	Approximately how much more time in hours, minutes, and seconds, Rocky estimates the task needs to complete.
Message	Additional information, including errors, pertaining to the task.

What would you like to do?

- Learn more About Panels (p. 78)
- Stop an Active Process (p. 1283)

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

5.6.6. Stop an Active Process

Close the window from which the process was originally requested. Note: If the processing window you want to close is dependent upon another request, you will be unable to close it until the dependency is cleared. Tip:Use the Progress panel to help you monitor the processing statuses of your various requests.

See Also:

- About the Progress Panel (p. 1281)

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information
of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

Chapter 6: Getting Help and Support

6.1. Help and Support

To get more help understanding the features and capabilities of Rocky, please consult the following resources.

Table 1: Product Documentation

Resource	Description	Location
Rocky User Manual (this resource)	Provides feature descriptions, parameter definitions, and step-by-step instructions for accomplishing various setup, processing, and post-processing tasks using the Rocky program's User Interface (UI). It is tied to the Rocky product version and receives updates with new software releases.	From within the Ansys Rocky program, click **Help**, point to **Manuals**, and then click **User Manual**.
Rocky SPH Technical Manual	Provides a brief description of the Smoothed-Particle Hydrodynamics (SPH) method and details the specific models and calculations used within the Ansys Rocky program.	From within the Ansys Rocky program, click **Help**, point to **Manuals**, and then click **SPH Technical Manual**.
Rocky Technical Manual	Provides a brief description of the Discrete Element Method (DEM) and details the specific models and calculations used within the Ansys Rocky program.	From within the Ansys Rocky program, click **Help**, point to **Manuals**, and then click **DEM Technical Manual**.
Rocky CFD Coupling Technical Manual	Provides best practices for using the Ansys 1- and 2-Way Fluent coupling methods within Rocky and explains some limitations of their usage.	From within the Ansys Rocky program, click **Help**, point to **Manuals**, and then click **CFD Coupling Technical Manual**.
Rocky PrePost scripting Manual	Lists the PrePost scripting functionality that can be used to create and modify Python *scripts* (p. 1394) for automating tasks in Rocky.	From within the Ansys Rocky program, click **Help**, point to **Manuals**, and then click **PrePost Scripting Manual**.
Rocky Solver SDK Manual	Lists the Solver SDK functionality that can be used to create and modify custom *external Modules* (p. 1391). Also includes instructions for using the SDK and setting up your programming environment, and also provides sample code you can use as examples.	From within the Rocky program, click **Help**, point to **Manuals**, and then click **Solver SDK Manual**.

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

Table 2: Training and Support

Resource	Description	Location
Ansys Customer Portal	Access self-guided Rocky workshops and installation instructions; download example scripts, modules, and the Calibration Suite; submit a customer support request or check-up on existing requests; and more.	Visit the support.ansys.com/portal/site/AnsysCustomerPortal page of the Ansys Customer Portal, and then sign in as instructed. If you do not already have sign-in information for the customer portal, please contact your Rocky representative for assistance.
Rocky Training	Sign-up for the next Rocky training session to get a better understanding of how Rocky can help you improve your designs, and get hands-on experience using the product.	Please contact your Rocky representative or Submit a Support Request (p. 1293) and ask for the latest training schedule.

What would you like to do?

- Submit a Support Request (p. 1293)
- Report a Software Bug (p. 1295)
- Include a Simulation Project in a Support Request (p. 1293)
- Verify Your Rocky License Status (p. 1295)

6.1.1. Ansys Engineering Copilot [BETA]

When the **Experimental (Beta) Features** checkbox is enabled on the **Options | Preferences** dialog (see also About Setting Global Preferences (p. 227)), this allows you to use the Ansys Engineering Co-pilot.

Important:

Note that beta features have not been fully tested and validated. Ansys, Inc. makes no commitment to resolve defects reported against these prototype features. However, your feedback will help us improve the overall quality of the product. We will not guarantee that the projects using this beta feature will run successfully when the feature is finally released so you may, therefore, need to modify the projects.

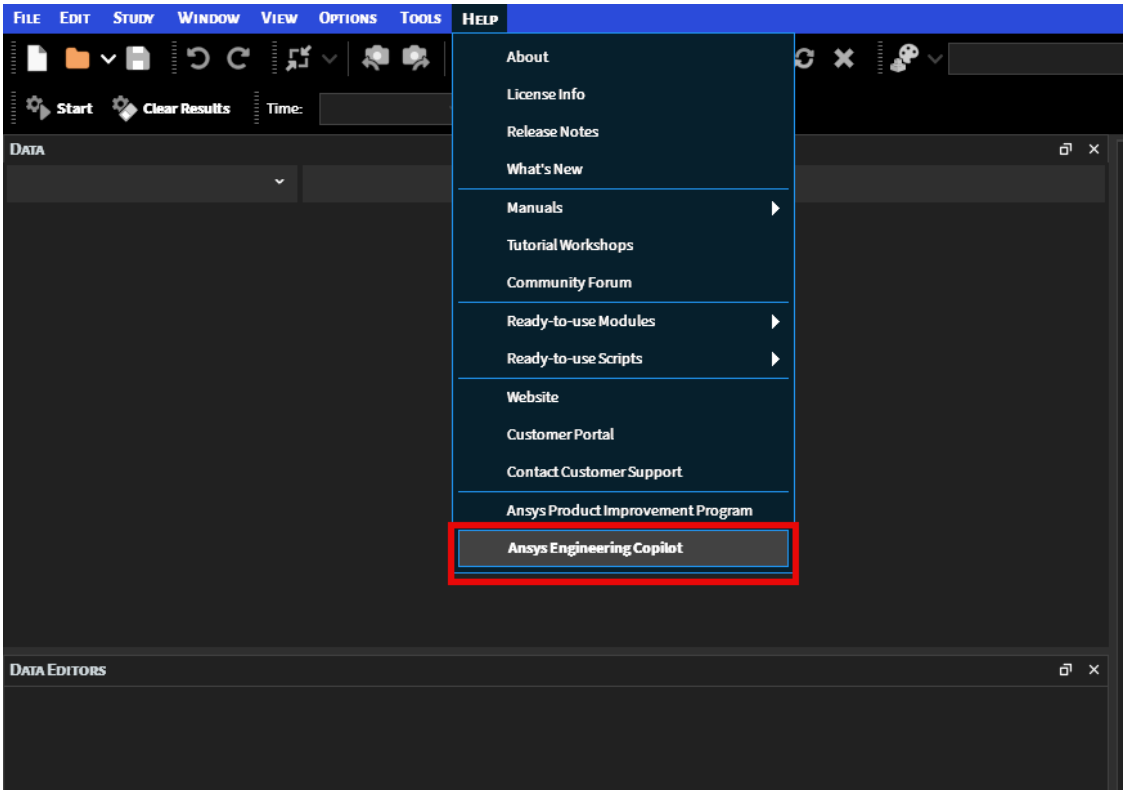
Ansys Engineering Copilot delivers content from Ansys content creators in response to your queries. Content sources that are searched include the Ansys Innovation Space, knowledge articles, forum posts, and **AnsysGPT™**.

Ansys Engineering Copilot opens a dock widget in Rocky software, where you can access several resources that will help you find answers to your questions and learn something new by engaging with the large global Ansys community. For more information, see the Ansys Engineering Copilot documentation.

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

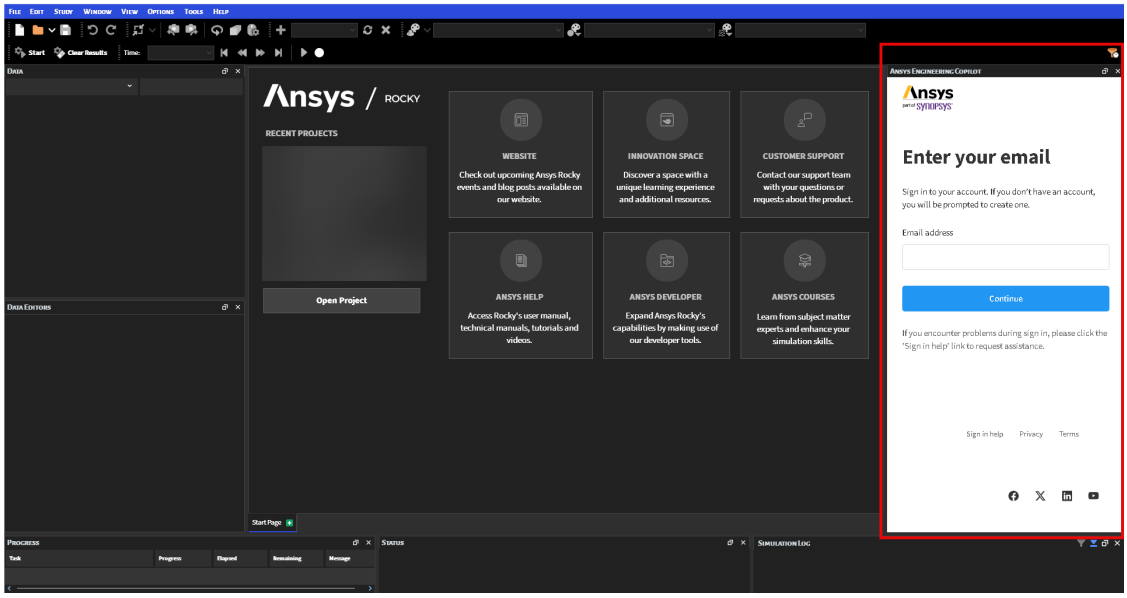
How to access Ansys Engineering Copilot in the Rocky software

- In the menu, click on **Help** > **Ansys Engineering Copilot**:

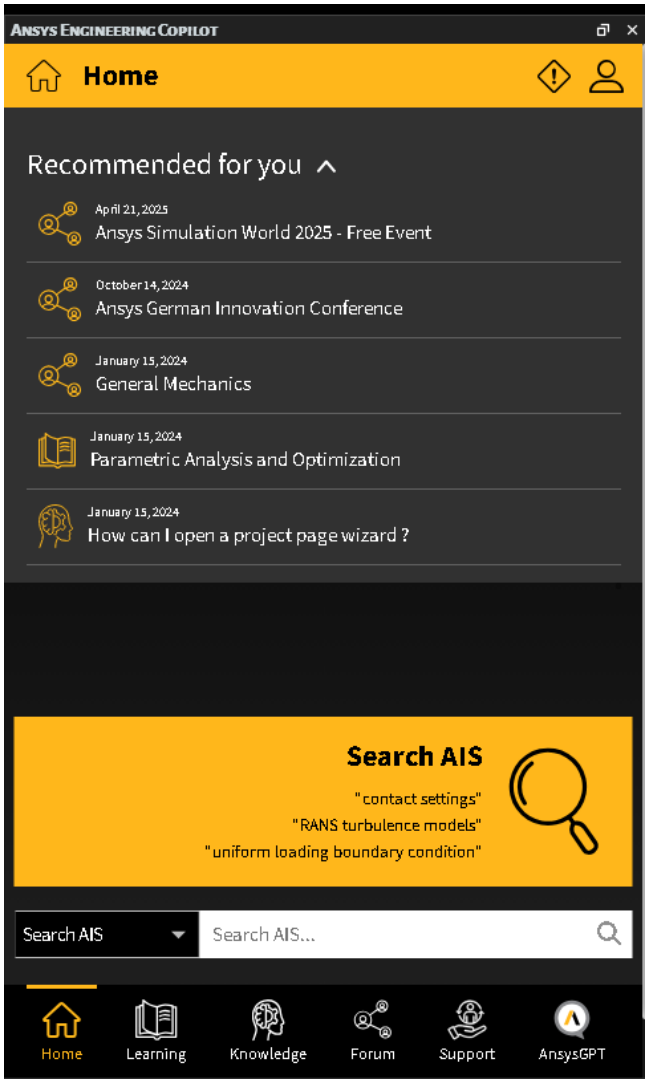


How to use

1. The first view of Ansys Engineering Copilot in a dock widget will look similar to the following:



2. Enter your credentials to sign in.



Pages

The Ansys Engineering Copilot dock widget contains the following pages:

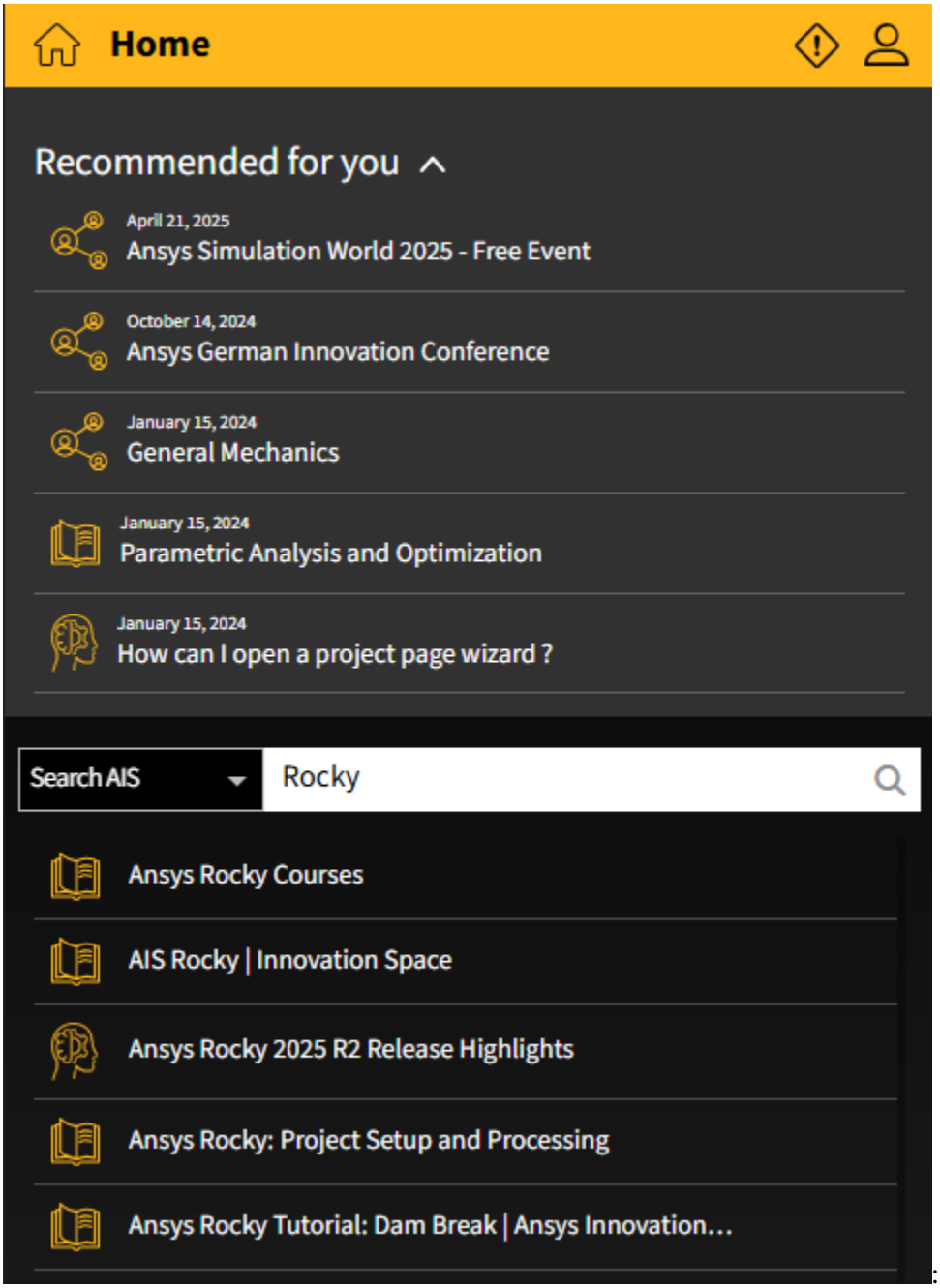


- **Home**

  This page suggests links that may be of interest to you.

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

In the search field, enter a topic and click the Search button


:

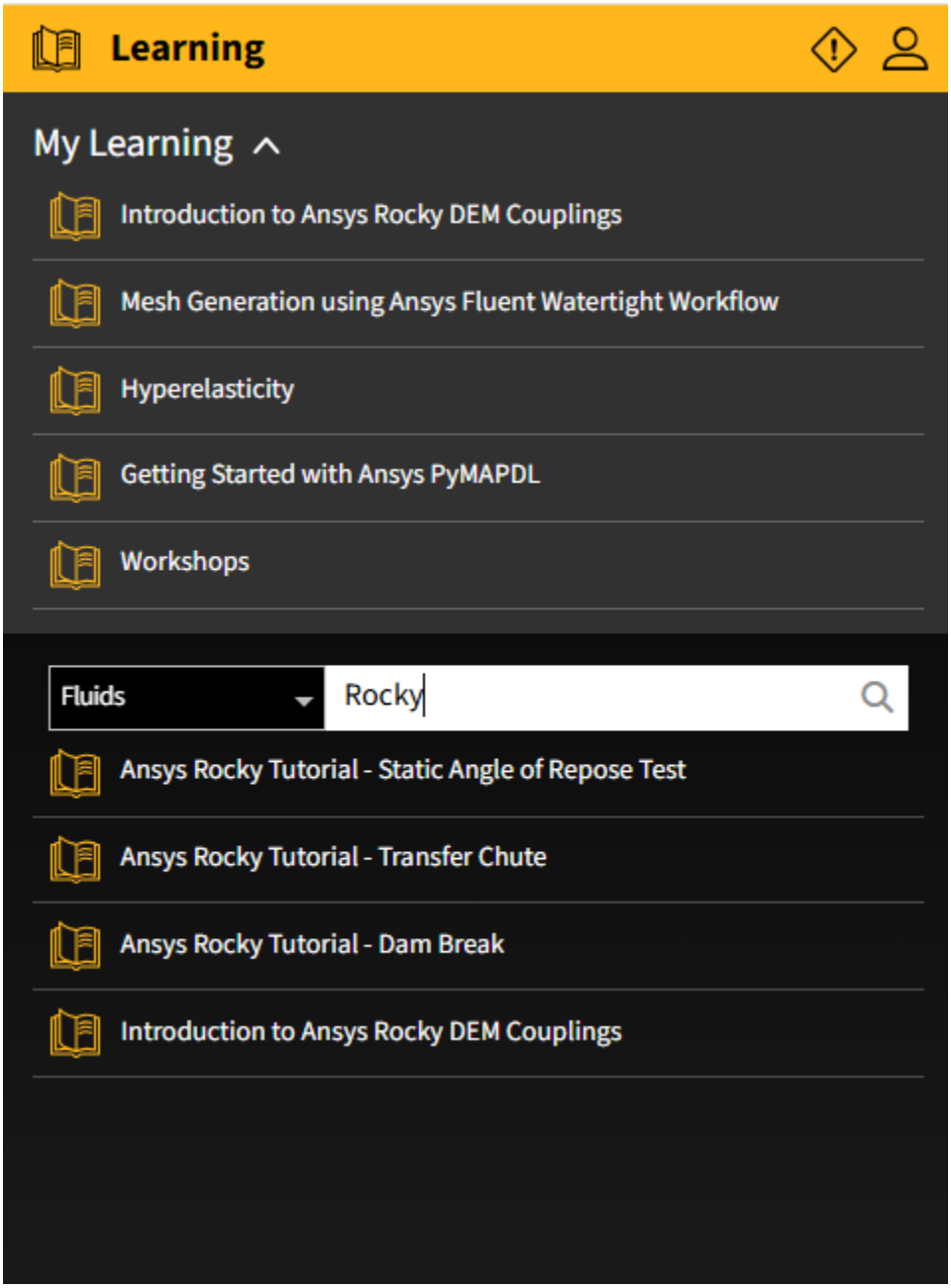
Click any of the search results to open.

- **Learning**

  This page lists any courses that have been assigned to you.

  Similar to the **Home** page, the **Learning** page also has a Search feature.

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

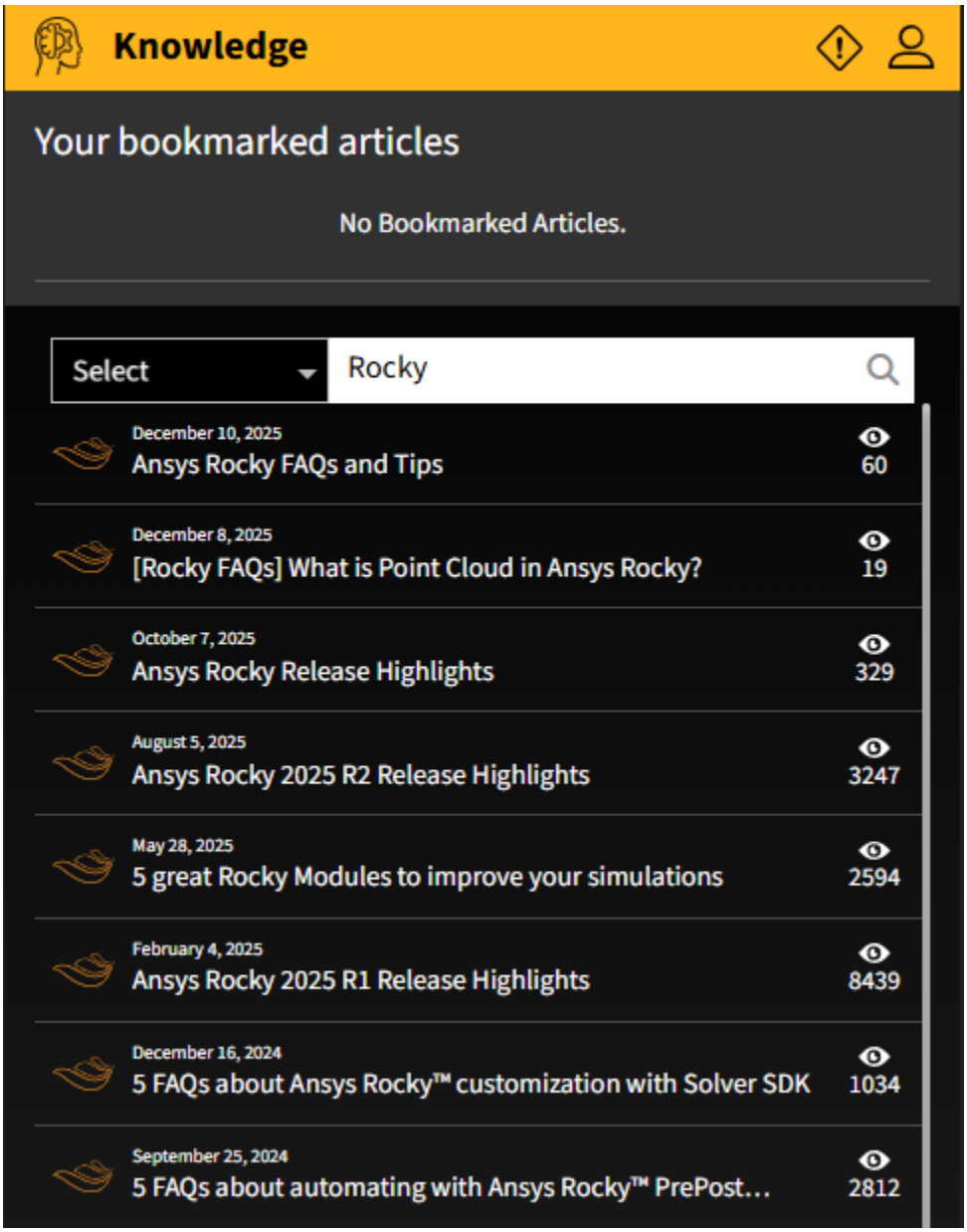


- **Knowledge**

  This page lists any pages you have bookmarked.

  You can also search for a topic. Unlike the Search feature on the **Home** page, clicking a search result here will open the page in the current dock widget. You can navigate back to the **Knowledge** page through the menu or using the "back" arrow in the UI.

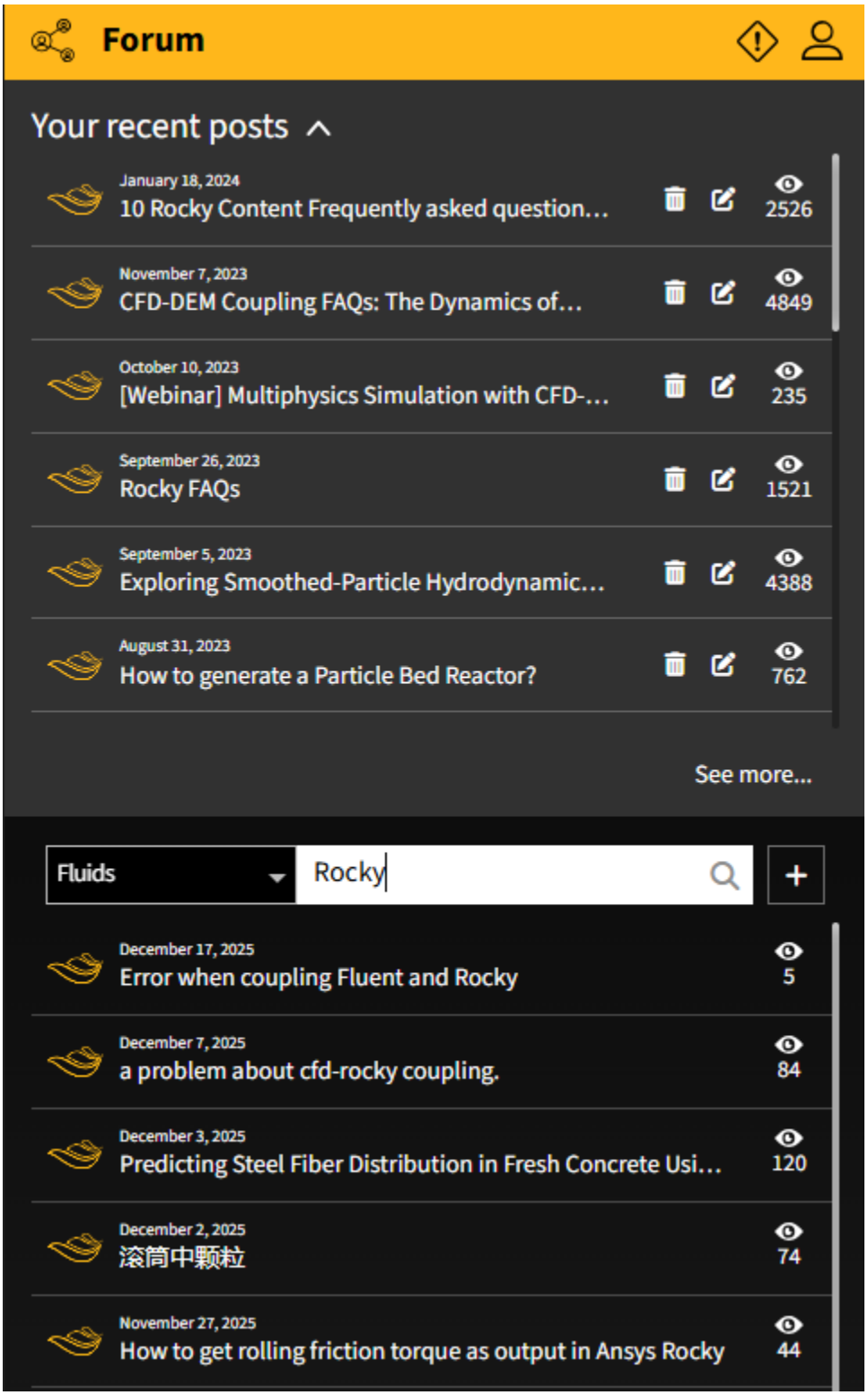
Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

- 

- **Forum**

  This page shows the recent forum pages you have visited. The **Forum** page also has a Search feature similar to that on the Knowledge page.

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.



- **Support**

  This page shows support information about your account.

- **AnsysGPT™**

  This page shows results from AnsysGPT™.

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

6.1.2. Submit a Support Request

The Rocky Support Team is available to help answer your questions about Rocky, provide assistance with licenses, report issues or mistakes in the software or documentation, help you troubleshoot issues with the software, and more.

To contact the Rocky Support Team, please access the Ansys Customer Portal.

Note:

If you do not already have sign-in information for the customer portal, please contact your Rocky representative for assistance.

This method should result in an automatic email being sent to your account, including the ticket number for your tracking purposes. A member of support will typically respond to your request within 1 business day. Once you receive the response, you may either continue to correspond to support through email, or through the customer portal.

Tip:

It can help provide context if you include an illustrative screenshot or even a copy of the simulation project your question is regarding. (See also Include a Simulation Project in a Support Request (p. 1293).)

See Also:

- Getting Help and Support (p. 1285)

6.1.3. Include a Simulation Project in a Support Request

When submitting a support request, it can help expedite the process if you provide a copy of the simulation project to which your request pertains. In this way, the support team can gain additional context as to your unique situation, and can provide answers appropriately.

Note:

If your project has already been processed and contains time steps, the associated files will likely be too large to send to support. It is therefore important to first save a copy (without results) of the project before you send it. See below for details.

To include a copy of your simulation project in a support request:

1. In Rocky, open the project you want to include with your support request.
2. Do one of the following:
   - If the project has already been processed (has time steps), follow the instructions to Save a Copy of Only the Simulation Settings; Not the Simulation Results (p. 854).

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

- If the project has not yet been processed (has no time steps), continue to step 3.

3. With your unprocessed project (no results) open, do one of the following:

   - To have Rocky automatically prepare an archive folder with all the files support needs, follow the instructions to Archive a Simulation Project (p. 1294).

   - To manually prepare your own folder with all the files support needs, Zip or 7zip the Rocky project (with no results), its associated project folder, and any custom particle .stl files you used in the project setup. Tip:The project folder will have the same prefix as the project file. *For Example*: If your project file is called "my_project.rocky", then you should zip both the "my_project.rocky" file and the "my_project.rocky.files" folder that is next to it.

4. Attach the .rocky_archive, Zip or 7zip folder to the ticket when submitting your support request. (See also Submit a Support Request (p. 1293).)

See Also:

- Getting Help and Support (p. 1285)

6.1.4. Archive a Simulation Project

Archiving a simulation project saves all the associated setup information, post-processing analyses, and results (time steps) for your project-including any scripts (p. 1394) you have saved to the **Project scripts** tab of the **PrePost Script** panel. (See also About Creating and Using PrePost Scripts (p. 296).)

For simulations that have already been processed, this can result in a very large file; therefore, the user has the possibility of archiving the project without the results. This will result in a much smaller file that can be more easily shared. (See also Include a Simulation Project in a Support Request (p. 1293).)

Note:

Depending upon the geometries and other setup criteria, archiving only the simulation setup portion of your Rocky project may still result in a large archive file, but should still be smaller than archiving the entire processed simulation.

To archive a simulation project:

1. From the **File** menu, click **Archive project**;

2. From the **Archive Options** select one between the options **Archive project with Simulation Results** or **Archive project WITHOUT Simulation Results;**

3. From the **Archive project** dialog, choose the drive or folder of the location to which you want to save the file, verify the **File name**, and then click **Save**;

4. From the **Project saved** message, click **Yes** to open the directory. The .rocky_archive file that you just saved is shown.

See Also:

- Include a Simulation Project in a Support Request (p. 1293)

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

- Open a Simulation Project (p. 411)
- Restore an Archived Simulation Project (p. 413)

6.1.5. Report a Software Bug

If you discover an issue or run into an unexpected error while using the Rocky software, please report it to Customer Support by Submitting a Support Request (p. 1293).

In your request, please include the following information:

- A short description of the behavior you are reporting.
- Step-by-step instructions for reproducing the behavior.
- Detail regarding the behavior you expected to see followed by the actual results you experienced.

See Also:

- Getting Help and Support (p. 1285)

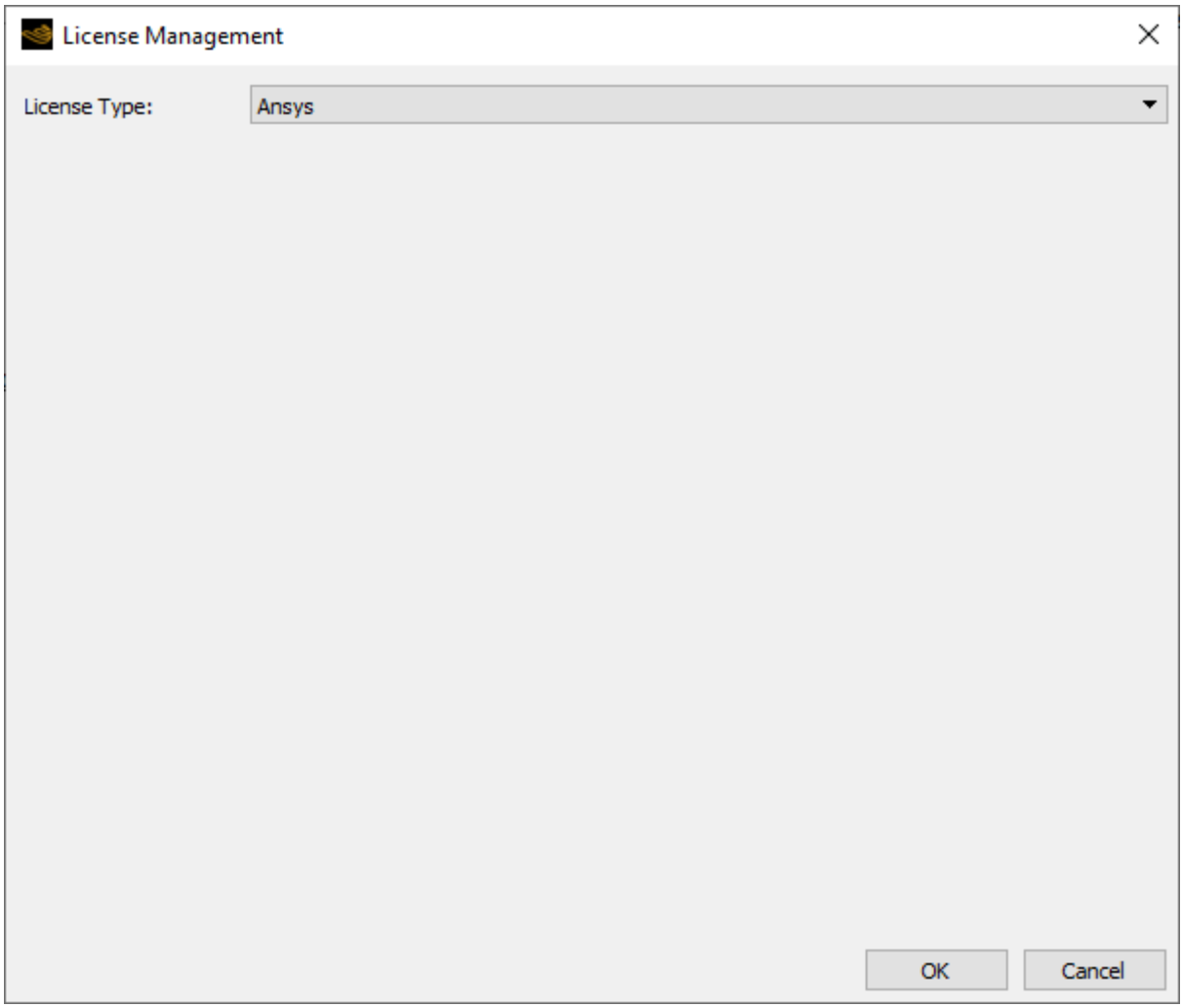
6.1.6. Verify Your Rocky License Status

1. From the Rocky **Help** menu, click **License Info**.

1. From the **License Management** dialog, look to the right of **License Type**. If your license is valid and set up correctly, you should see **Ansys** listed (Figure 1).

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

Figure 6.1: License Management dialog showing a License Type of Ansys



See Also:

- My Rocky license Status says "No license for feature" (p. 1380)
- Getting Help and Support (p. 1285)

6.2. Remove Rocky

Instructions for removing Rocky on both Windows- and Linux-based platforms are provided below.

Tip: For a clean install of Rocky, refer to the I reinstalled Rocky but I am still seeing errors (p. 1384) topic.

6.2.1. To remove Rocky [Windows]

1. From the Programs list on the Windows **Start** menu, click **Ansys**, right-click **Rocky 2023 R1**, and then click **Uninstall**.
2. Follow the instructions provided on your screen.

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

6.2.2. To remove Rocky [Linux]:

Manually find and then remove the Rocky installation folder from your machine.

See Also:

- Getting Started (p. 73)
- Getting Help and Support (p. 1285)
- Uninstall Ansys Coupling Components (p. 795)
- I reinstalled Rocky but I am still seeing errors (p. 1384)

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information
of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

Chapter 7: Appendices

This section contains supplemental information that might assist you with using some of Rocky's more advanced features.

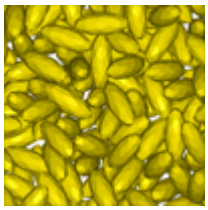
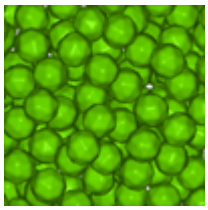
What would you like to review?

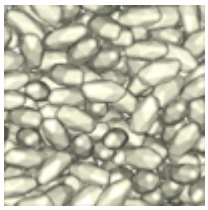
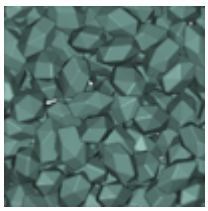
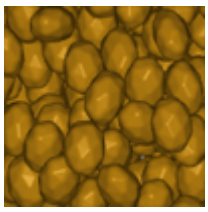
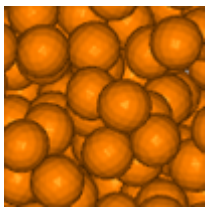
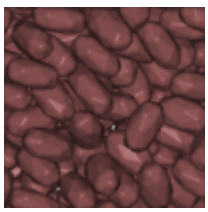
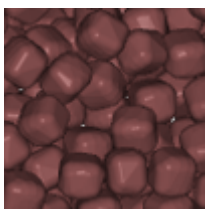
- Appendix A: Particle Shape Examples (p. 1299)
- Appendix B: ASCII Printable Characters (p. 1304)
- Appendix F: Save Geometries as XGL, STL, or DXF Files (p. 1307)

7.1. Appendix A: Particle Shape Examples

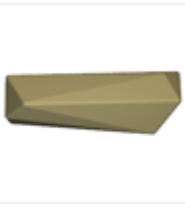
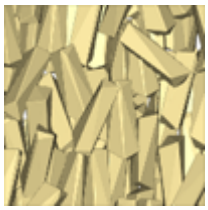
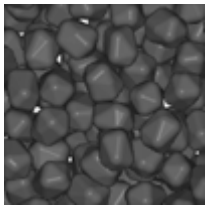
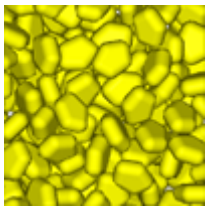
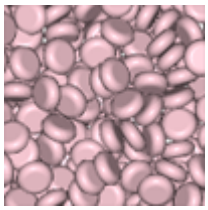
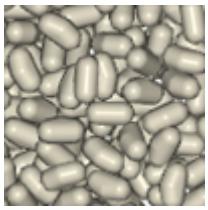
Use the following examples to help you create the perfect particle shape for your project. The examples below make use of the **Solid** particle shapes provided by default in Rocky. If the default shapes aren't enough to meet your needs, you may also import your own custom particle shape. (See also Import Custom Particle Shapes (p. 677).)

Table 1: Default Solid Particle Shape Examples

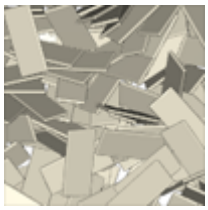
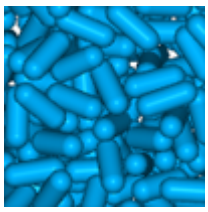
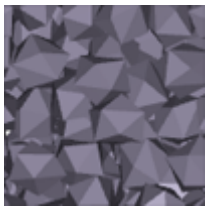
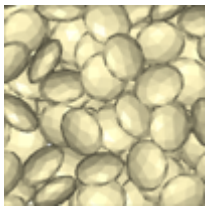
Start with this Particle Type	To create this shape		Vertical Aspect Ratio	Horizontal Aspect Ratio	Smoothness	Side Angle	Number of Corners	Superqua Degree
			2.70	1.00	0.16	-	48	2.00
Sphero-Polyhedron	Barley							
			0.92	1.00	0.96	-	25	2.60
	Pea							

Start with this Particle Type	To create this shape		Vertical Aspect Ratio	Horizontal Aspect Ratio	Smoothness	Side Angle	Number of Corners	Superqua Degree
	 Rice		2.70	0.70	0.96	-	16	2.00
	 Rock		1.50	0.75	0.25	-	15	2.00
	 Potato		0.70	1.70	1.00	-	27	2.00
	 Orange		0.90	1.00	1.00	-	104	2.00
	 Raisin		2.30	0.70	1.00	-	22	2.80
	 Chocolate		1.00	0.79	1.00	-	40	10.50

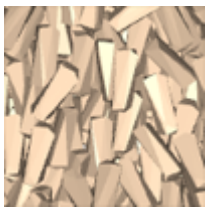
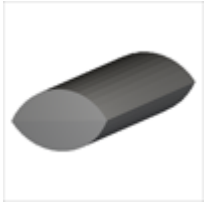
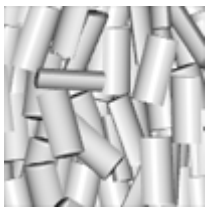
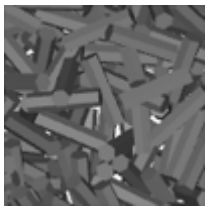
Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

Start with this Particle Type	To create this shape		Vertical Aspect Ratio	Horizontal Aspect Ratio	Smoothness	Side Angle	Number of Corners	Superqua Degree
	 Wood Chip		0.70	2.70	0.10	-	10	12.00
	 Coal		1.25	0.70	1.00	-	11	6.20
Start with this Particle Type	To create this shape		Vertical Aspect Ratio	Horizontal Aspect Ratio	Smoothness	Side Angle	Number of Corners	Superquad Degree
 Sphero-Polygon	 Poker Chip		1.05	0.10	-	-	32	-
	 Corn		1.24	0.53	-	-	5	-
	 Pill		1.00	0.45	-	-	40	-
	 Bean		1.80	0.85	-	-	3	-

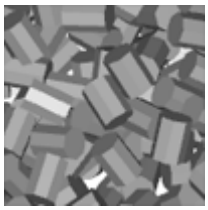
Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

Start with this Particle Type	To create this shape		Vertical Aspect Ratio	Horizontal Aspect Ratio	Smoothness	Side Angle	Number of Corners	Superqua Degree
			2.37	0.10	-	-	4	-
	Sheet Metal							
Start with this Particle Type	To create this shape		Vertical Aspect Ratio	Horizontal Aspect Ratio	Smoothness	Side Angle	Number of Corners	Superquad Degree
			9.41	-	-	-	-	-
Sphero-Cylinder	Smooth Rod							
			2.57	-	-	-	-	-
	Capsule							
Start with this Particle Type		To create this shape	Vertical Aspect Ratio	Horizontal Aspect Ratio	Smoothness	Side Angle	Number of Corners	Superquad Degree
			1.07	0.92	-	-	10	2.00
Polyhedron	Sharp Rock							
			1.04	0.50	-	-	57	2.00
	Wafer							

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

Start with this Particle Type	To create this shape		Vertical Aspect Ratio	Horizontal Aspect Ratio	Smoothness	Side Angle	Number of Corners	Superqua Degree
	 Brick		0.50	1.76	-	-	44	12.00
	 Firewood		0.70	2.50	-	-	14	12.00
Start with this Particle Type	To create this shape		Vertical Aspect Ratio	Horizontal Aspect Ratio	Smoothness	Side Angle	Number of Corners	Superquac Degree
 Briquette	 Newspaper		0.60	2.01	-	5.00	60	-
	 Package		0.30	1.45	-	34.50	32	-
Start with this Particle Type	To create this shape		Vertical Aspect Ratio	Horizontal Aspect Ratio	Smoothness	Side Angle	Number of Corners	Superquac Degree
 Faceted-Cylinder	 Hex Rod		3.75	-	-	-	12	-

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

Start with this Particle Type	To create this shape		Vertical Aspect Ratio	Horizontal Aspect Ratio	Smoothness	Side Angle	Number of Corners	Superqua Degree
	 Octagon Socket		1.30	-	-	-	16	-

7.2. Appendix B: ASCII Printable Characters

The below chart for the ASCII printable characters (codes 32-127) is useful for understanding how sort order is determined in Rocky. (See also Name Components to Affect Sort Order (p. 112)).

Symbol	Description
	Space
!	Exclamation mark
"	Double quotes (or speech marks)
#	Number
$	Dollar
%	Percentage
&	Ampersand
'	Single quote
(	Open parenthesis (or open bracket)
)	Close parenthesis (or close bracket)
*	Asterisk
	Plus
,	Comma
	Hyphen
.	Period, dot, or full stop
/	Slash or divide
0	Zero
1	One
2	Two
3	Three
4	Four

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

Symbol	Description
5	Five
6	Six
7	Seven
8	Eight
9	Nine
:	Colon
;	Semicolon
<	Less than (or open angled bracket)
=	Equals
>	Greater than (or close angled bracket)
?	Question mark
@	At symbol
A	Uppercase A
B	Uppercase B
C	Uppercase C
D	Uppercase D
E	Uppercase E
F	Uppercase F
G	Uppercase G
H	Uppercase H
I	Uppercase I
J	Uppercase J
K	Uppercase K
L	Uppercase L
M	Uppercase M
N	Uppercase N
O	Uppercase O
P	Uppercase P
Q	Uppercase Q
R	Uppercase R
S	Uppercase S
T	Uppercase T
U	Uppercase U
V	Uppercase V
W	Uppercase W
X	Uppercase X

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

Symbol	Description
Y	Uppercase Y
Z	Uppercase Z
[	Opening bracket
\	Backslash
]	Closing bracket
^	Caret - circumflex
_	Underscore
`	Grave accent
a	Lowercase a
b	Lowercase b
c	Lowercase c
d	Lowercase d
e	Lowercase e
f	Lowercase f
g	Lowercase g
h	Lowercase h
i	Lowercase i
j	Lowercase j
k	Lowercase k
l	Lowercase l
m	Lowercase m
n	Lowercase n
o	Lowercase o
p	Lowercase p
q	Lowercase q
r	Lowercase r
s	Lowercase s
t	Lowercase t
u	Lowercase u
v	Lowercase v
w	Lowercase w
x	Lowercase x
y	Lowercase y
z	Lowercase z
{	Opening brace
|	Vertical bar
}	Closing brace
~	Equivalency sign - tilde

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

Symbol	Description
	Delete

7.3. Appendix F: Save Geometries as XGL, STL, or DXF Files

Important:

This section contains procedures that occur outside of this version of Rocky. This information is provided as supplemental only; when third-party products are referenced, the content provided is not meant to replace the procedures or documentation included within those third-party products. Refer to the documentation that came with your third-party product for additional information.

Rocky supports importing geometries that have been saved as XGL, STL, or DXF 3D faces files. From within a CAD program, such as Ansys SpaceClaim, you might need to **Save As Copy** or **Export** your geometries to find these file types.

Note: DXF 3D faces is a format that might only be available in older versions of CAD programs. See your CAD program's user documentation for DXF support and export information.

What would you like to do?

- Use Ansys SpaceClaim 2022 R2 to Save Geometries to STL Files (p. 1307)

7.3.1. Use Ansys SpaceClaim 2022 R2 to Save Geometries to STL Files

1. From the SpaceClaim **File** menu, point to **Save As**, and then click **Save As**.
2. From the **Save as type** list, click **STL (*.stl)**, and then enter a name and location for the file.
3. **Optional**: Do any or none of the following:
   - By default, SpaceClaim saves the file using the same unit with which you are working (for example, mm). If you want to save using a different unit (for example, meters), you have to enable the **Override Units** checkbox, and then select your desired unit from the units box.
   - If you want to save one file per component, instead of a single file containing all the geometry parts, click **Options** and then from the **STL file options** panel, select **Per component** from the **Export** tab. Saving your geometry as one file per component can be useful in cases where you want to prescribe different materials interactions coefficients to different parts, or different motion frames to different parts, and so on.
   - Also on the **Export** tab, you can increase the geometry resolution by changing the **Resolution** from **Medium** to **Fine**. Note that by choosing to use smaller triangles, the number of triangles making up the geometry and therefore the computational cost will be increased.
4. Click **Save**.

See Also:

- About Importing Geometry Components (p. 487)

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

- About Defining and Importing Custom Particle Shapes (p. 670)

7.4. Appendix G: Perform Tasks from the Command Line or Journal

If you need to perform Rocky tasks without using the Rocky UI, also known as running in batch mode, it is possible to do so from the command line on both Windows- and Linux-based machines.

You can also perform batch Fluent runs by using a journal.

7.4.1. About Using Commands

For example, you can process your Rocky simulation project without ever seeing the Rocky UI (sometimes known as hidden mode or silent mode) by passing `--simulate` and the path and name of the project file you want processed, along with other parameters such as whether you want the simulation processed on CPU or GPU resources.

It is important to note that the Rocky executable exists inside different folders on different operating systems. Specifically:

- In Windows, the `Rocky.exe` file exists inside the `bin` folder, the standard location being `C:/Program Files/ANSYS Inc/v241/Rocky/bin`
- In Linux, the `Rocky` file exists in no standard location; however, the location might look something like `<Path to Rocky Installation>`

7.4.2. Known Limitations

There are some situations where using Rocky commands in batch mode will not work. See the following sections for more information.

Scripts that Require the Rocky UI to be Open

You might get errors in Rocky (or in Ansys Workbench) when a script for the Rocky simulation (or Workbench project) you are running in batch mode includes tasks that require the Rocky UI to be open. These tasks include:

- Creating and exporting an animation.
- Creating and exporting plots.
- Creating and exporting images of 3D View windows.

See also I get a scripting error when I attempt to run the simulation in batch mode (p. 1349).

Scripts that use matplotlib

You might get errors in Rocky (or in Ansys Workbench) when a script for the Rocky simulation (or Workbench project) you are running in batch mode makes use of matplotlib using Qt as a backend. This can happen if the script tries to export an image, for example.

See also I get a "Server returned a fault exception" error when running a script in batch mode (p. 1350).

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

7.4.3. Common Commands

See the table and procedures listed below to learn more about using commands to perform Rocky tasks.

Table 1: Rocky Command Line Parameters

Command	Definition	Range
Processing Commands		
--simulate	Processes from the beginning, in hidden mode, the project file name and location that follows in quotes.	<file path to project `project_file-name.rocky`>
--ncpus	Sets the number of CPU processors on which the simulation will be run.	Positive whole numbers
--resume	Resumes an in-process simulation, if possible.	0 = Off 1 = On
--use-gpu	Processes the simulation on a CUDA-enabled GPU.	0 = Off 1 = On
--gpu-num	The index of the GPU to use for processing. **Note:** This setting may appear multiple times to run in multiple GPUs. **Tip:** To determine which GPU maps to which index, open the Rocky UI and look at the **Target GPU** list in the **Solver** tab. The index is based upon the order shown, starting with 0.	Positive whole numbers
Other Commands		
--script	Runs a script (p. 1394) .py file, the name of which follows in quotes.	`<file path to script script_file-name.py>`
--headless	Process from the beginning, in hidden mode, the script (p. 1394) .py file name of which follows in quotes.	`<file path to script`

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

Command	Definition	Range
		`script_file-name.py>`
MPI (Message Passaging Interface) Commands (BETA)	**Important:** Note that beta features have not been fully tested and validated. Ansys, Inc. makes no commitment to resolve defects reported against these prototype features. However, your feedback will help us improve the overall quality of the product. We will not guarantee that the projects using this beta feature will run successfully when the feature is finally released so you may, therefore, need to modify the projects.	
--use-mpi	Activate the MPI parallelism.	
--mpi-nprocesses	Set the number of MPI processes to use on the simulation.	One process for each CPU (Windows and Linux support) or GPU (only Linux support).
--mpi-ncpus-per-process	Set the number of CPUs' cores per solver process.	From 1 to maximum CPU cores avaialble on your CPU unit.
--mpi-hosts	Comma-separated list of hostnames or IPs to be used by MPI. (Not necessary if you use a scheduler).	All your system names or IP addresses.
--mpi-hostfile	Path to a text file that describes the hostnames and number of processes per host. (Not necessary if you use a scheduler). **Note:** Use *--mpi-hostfile* or *--mpi-hosts*.	All your system names or IP addresses.
--mpi-extra-params	Extra parameters to be passed to the MPI command line.	

Important:

For CFD Coupling simulations using the 2-Way Fluent method, items on the **2-Way Fluent | Fluent** tab, (i.e., **Execution mode**, **Solver Processes**, and so on) will be controlled by Rocky and should therefore be defined inside the Rocky project. These parameters can be

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

used to perform Rocky - Fluent tasks from the command line. (See also About Using the 2-Way Fluent Method (p. 795).)

What would you like to do?

- Use the Command Line to Process a Simulation (p. 1311)
- Use the Command Line to Play a Script (p. 1313)
- Use a Journal to Run Fluent and Export Transient 1-Way Results (p. 1313)

7.4.4. Use the Command Line to Process a Simulation

1. Open a command prompt.
2. Using commands, navigate to the directory containing the Rocky executable. For example:
   - In Windows, the standard location for the `Rocky.exe` file is `C:\Program Files\ANSYS Inc\v241\Rocky\bin`
   - In Linux, there is no standard but the location for the `Rocky` file might best be described as `<Path to Rocky Installation>`
3. Type the name of the executable (`Rocky` in Linux and `Rocky.exe` in Windows).followed by `--simulate`, followed by the file name and location you want to process in quotes, followed by any other parameters you want to define. (See also Appendix G: Perform Tasks from the Command Line or Journal (p. 1308).) Windows Examples:
   - Command to run with multiple GPUs:

     ```
     Rocky.exe --simulate

     "C:Documents\project_filename.rocky"

     --resume=0 --use-gpu=1 --gpu-num=0 --gpu-num=1
     ```

   - Command to resume a simulation with multiple CPUs:

     ```
     Rocky.exe

     --simulate "C:\Documents\project_filename.rocky"

     --resume=1 --ncpus=7
     ```

   - Command to select multi gpu slicing direction (X,Y or Z):

     ```
     Rocky.exe --simulate

     "C:\Documents\project_filename.rocky"

     --multi-gpu-slicing-direction=X
     ```

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

4. Type the name of the executable (Rocky in Linux and Rocky.exe in Windows),followed by `--script`, followed by the script file name and the location you want to process in quotes; followed by `--headless`, followed by any other parameters you want to define.

For example:

- Command to run a script in batch mode:

```
Rocky.exe --script "C:\Documents\project_filename.py" --headless
```

Your simulation starts processing in the background.

Tip:

Because you are running this simulation in batch mode, you can verify if Rocky has opened the simulation to start processing it by looking for the presence of the `.lock` file in the project file directory. If this file is present along with for your `.rocky` project file, then Rocky has likely opened and started processing it in the background. (See also Rocky says my project file is "locked" (p. 1382).)

Additional Tips:

- Because it can be hard to track the progress of your simulation if there is no user interface open, you can choose to have a file be written containing information about the previous commands by using a pipe redirection, i.e `> log.txt`.

  For example, in Windows, this might be:

  `Rocky.exe --simulate "C:\Documents\project_filename.rocky" > C:\Documents\log.txt`.

  If you want to review the information more frequently, you can use the:

  `tail -f <path to log file>` (on Linux) or the `type -f <path to log file>` (on Windows).

  which will freeze the terminal and output the last content that was written to the file specified.

- If you need to stop Rocky from processing the project, do one of the following:

  - On Linux, press `Ctrl` + `C` on your keyboard.

  - On Windows, open the Task Manager, and then end the `RockySolver.exe` process.

- Since there is no default folder for installation in Linux, there are two main ways you can run Rocky by using a simple, one-word command:

  – **Define and use an alias**: This allows you to use a different (shorter) name for the executable.

  Example: Assuming you have installed Rocky at `/home/user/Ansys/Rocky`, enter `alias Rocky=/home/user/Ansys/Rocky/Rocky`.

  Then, you are able to simply use `Rocky --simulate <project>`.-

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

- **Edit the PATH variable**: This allows you to add a folder to the list of places Linux will look for the executable.

  Example: Assuming you have installed Rocky at `/home/user/Ansys/Rocky`, enter

  `export PATH=$PATH:/home/user/Ansys/Rocky`.

  Then, you are able to simply use `Rocky --simulate <project>`.

Both of these changes are temporary so you should consider storing those lines on .bashrc or equivalent to have them persist during the changing of your terminal or during reboots.

See Also:

- Processing a Simulation (p. 907)
- Appendix G: Perform Tasks from the Command Line or Journal (p. 1308)

7.4.5. Use the Command Line to Play a Script

1. Open a command prompt.
2. Using commands, navigate to the directory containing the Rocky executable. For example:
   - In Windows, the standard location for the `Rocky.exe` file is `C:Program FilesANSYS Incv222Rockybin`
   - In Linux, there is no standard but the location for the `Rocky` file might best be described as `<Path to Rocky Installation>`
3. Type the name of the executable (`Rocky` in Linux and `Rocky.exe` in Windows), followed by the .rocky file name and location of the project you want to open in quotes, followed by `-script` followed by the PY file name and location of the script you want to apply, also in quotes. Windows Example: `Rocky.exe "C:Documentsproject_filename.rocky" -script "C:DocumentsRockyScriptsscript_filename.py"`

The Rocky user interface opens the indicated project and attempts to run the script.

Note: If `-script` is used together with `-simulate`, the script will attempt to play but might fail if UI elements, such as Camera Presets, are required for completing the recorded or scripted steps.

See Also:

- About Creating and Using PrePost Scripts (p. 296)
- Appendix G: Perform Tasks from the Command Line or Journal (p. 1308)

7.4.6. Use a Journal to Run Fluent and Export Transient 1-Way Results

Use the following information to run Fluent and export the results for a 1-Way transient simulation using a journal, a process that is used for batch Fluent runs. This will enable you to run your CFD case and export the results without opening the Fluent UI.

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

```
/file/start-transcript "fluent.log"

(define rocky/library-path "

{lib_directory}") (ti-menu-load-string "/file/read-macros "{lib_directory}

rocky_fluent_coupling.scm""))

/file/read-case "

{base_cas_filename}

" /solve/set/time-step 0.0005

(set-f2r-folder "

{f2r_folder}

")

/solve/initialize/initialize-flow (one-way-transient-export)

(display (rpgetvar 'rocky/additional-output-name-list)) (display (rpgetvar
'rocky/additional-output-phase-id-list)) (display (rpgetvar 'rocky/addi-
tional-output-id-list)) (display (rpgetvar 'rocky/additional-output-type-
list))

/solve/dual-time-iterate 2 2 (stop-one-way-transient-export)

exit OK
```

See Also

- About Using the 1-Way Fluent Method (p. 775)
- Use the 1-Way Fluent Method to Process Fluent and Rocky Simulations (p. 383)
- Appendix G: Perform Tasks from the Command Line or Journal (p. 1308)

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

Chapter 8: Troubleshooting

This section contains solutions to some common Rocky user issues. If your problem is not solved in this section, see Getting Help and Support (p. 1285) for additional assistance.

8.1. What kind of problems are you having?

- Simulation Setup Problems (p. 1315)
- Simulation Processing Problems (p. 1338)
- Simulation Analysis Problems (p. 1363)
- Rocky Licensing and File Problems (p. 1379)

See Also:

- Getting Help and Support (p. 1285)

8.2. Troubleshooting Simulation Setup Problems

What kind of problem are you having?

- I cannot change my import scale (p. 1316)
- I cannot change the setup parameters (p. 1317)
- I cannot find PrePost Scripting documentation (p. 1317)
- I cannot find Solver SDK documentation (p. 1318)
- I cannot get Rocky to couple with my CFD program (p. 1319)
- I am having difficulty properly calibrating sticky, wet, or cohesive particles (p. 1319)
- I do not know how to simulate magnetic forces (p. 1319)
- I need help simulating an HPGR (p. 1319)
- I cannot find the upper limit for importing custom particles shapes (p. 1320)
- I entered an invalid variable or value and Rocky accepted it (p. 1320)
- I briefly see long numbers when pasting in copied values (p. 1323)
- My particle size is too large for my periodic domain (p. 1324)

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

- Parts of my particle disappear or appear cut off (p. 1324)
- Rocky does not list my Ansys version when I try to set up 2-Way Fluent Coupling (p. 1326)
- I cannot find the Rocky component in Workbench (p. 1327)
- I get "reference" errors in my Motion Frames when I open an older project (p. 1327)
- I cannot find the Forces for FEM Analysis checkbox (p. 1329)
- I get an "Unsupported Particle" error when I open an older project (p. 1329)
- I cannot find Particle Collision Statistics (p. 1330)
- I cannot find Particle Size Scale Factor (p. 1330)
- I cannot find the Leeds Adhesion Force model (p. 1331)
- I cannot find the Inlet Start and Stop Times (p. 1331)
- I Cannot Find the Movements Tab for My Older Simulation Projects (p. 1331)
- I cannot figure out how to set the Parent Reference for a Motion Frame (p. 1332)
- I get "Displacement" errors in my Motion Frames when I open an older project (p. 1333)
- My "Air Viscosity" units changed (p. 1334)
- I cannot find the Velocity Dependent Adhesive Force model (p. 1335)
- I cannot enter an input variable or mathematical function into a text field (p. 1334)
- I cannot find the Torque Law setting for 1-Way Constant (p. 1336)
- I cannot find the Output Frequency parameter (p. 1336)
- I cannot find the Loading N-Steps parameter (p. 1336)
- I cannot find the Set Contact Detection Neighboring Distance parameter (p. 1337)
- I cannot find the Export Solution to Rocky item in Fluent (p. 1337)
- I cannot find the Maximum Volume Fraction Target parameter for 2-Way Fluent (p. 1338)

See Also:

- Troubleshooting (p. 1315)

8.2.1. I cannot change my import scale

The import scale cannot be changed once the geometry files have been imported. To change the scale, you must replace the geometry file and then change the scale on re-import. (See also Replace an Imported Geometry File (p. 521).)

See Also:

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

- About Importing Geometry Components (p. 487)

8.2.2. I cannot change the setup parameters

If you are unable to change the setup parameters of your project, then you have most likely already begun to process the simulation. Once you click **Start** on the **Solver** tab, you cannot change the setup parameters without losing the simulation results that have already been calculated. This is because the setup parameters directly affect the calculations that Rocky is solving.

If you must change the parameters after you have started processing the simulation, first stop the simulation (see also Stop a Simulation from Processing (p. 915)), change your setup parameters, and then start your simulation over from the beginning (see also Start Processing a Simulation from the Beginning (p. 913).).

See Also:

- About Starting a Simulation (p. 908)

8.2.3. I cannot find PrePost Scripting documentation

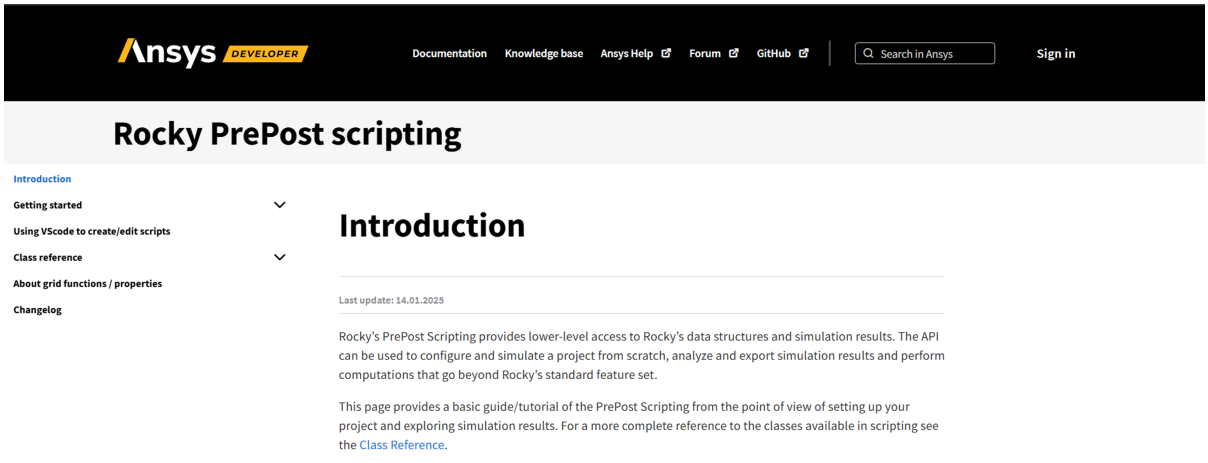
PrePost Scripting allows you to create PrePost Scripts (p. 1394) for the purpose of automating many setup and post-processing tasks.

Information about PrePost Scripting functionality can be found in two places: in the PrePost Scripting Manual and in the Python Shell.

PrePost scripting Manual

Setup and post-processing Classes and Methods are documented in the PrePost Scripting Manual (Figure 8.1: PrePost Scripting Manual (p. 1317)). (From the Rocky **Help** menu, point to **Manuals** and then click **PrePost Scripting Manual**).

Figure 8.1: PrePost Scripting Manual



Once the PrePost Scripting Manual opens, use the **Index** or **Search** bar to find the Classes and Methods you need.

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

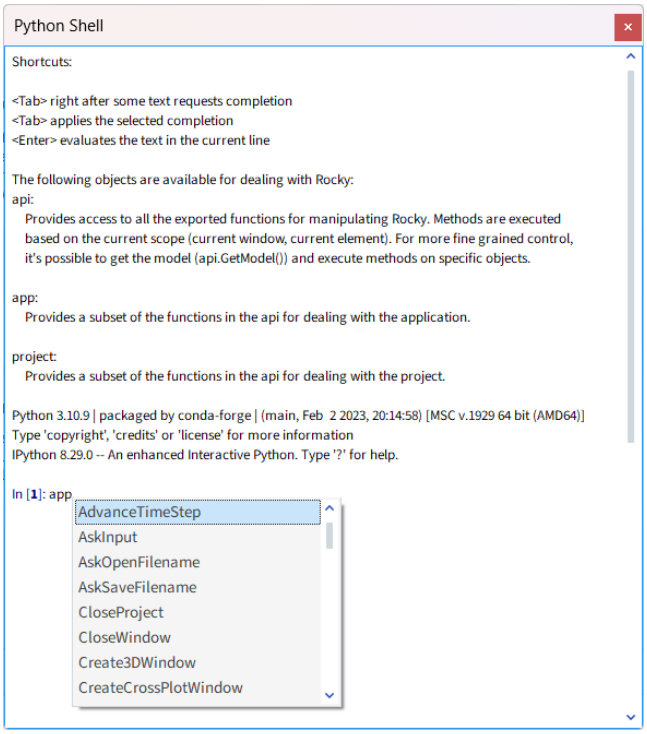
Python Shell

The PrePost Scripting functionality documented in the PrePost Scripting Manual is available in the Python Shell panel. (See also Customize and Extend Rocky with Python (p. 294).)

For example, type **app.** into the Python Shell panel and then press **Tab** to see a list of what application-level PrePost APIs are available (Figure 8.2: List of application-level PrePost Scripting functionality available for use in your PrePost Scripts (p. 1318)).

If you need additional assistance creating your PrePost Scripts, please Submit a Support Request (p. 1293).

Figure 8.2: List of application-level PrePost Scripting functionality available for use in your PrePost Scripts



8.2.4. I cannot find Solver SDK documentation

Solver SDK allows you to use the Rocky Software Development Kit (SDK) to make your own custom modules.

Information about Solver SDK functionality can be found in the Solver SDK Manual. (From the Rocky **Help** menu, point to **Manuals**, and then click **Solver SDK Manual**.)

See Also:

- Customize and Extend Rocky with Python (p. 294)
- About Creating and Using PrePost Scripts (p. 296)
- About Modules Parameters (p. 431)

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

8.2.5. I cannot get Rocky to couple with my CFD program

Currently, Ansys Fluent is the only program supported by Rocky for CFD Coupling purposes. This is especially true of Two Way Coupling. (See also About Using the 2-Way Fluent Method (p. 795).)

However, if your CFD program is configured to the same format as Fluent, it might be possible for Rocky to accept another program's CFD data during a 1-Way Fluent process. (See also About Using the 1-Way Fluent Method (p. 775).)

See Also:

- Set or Modify Fluid and/or Air Flow Properties (p. 757)

8.2.6. I am having difficulty properly calibrating sticky, wet, or cohesive particles

Sticky, wet, and otherwise especially cohesive particles can be difficult to properly calibrate.

You may wish to use the Material Wizard and/or the Calibration Suite to help you in this process. These automated solutions can help you quickly and easily determine more accurate DEM parameters to more realistically represent your target materials.

If you are interested in learning more about how the particle-to-particle and particle-to-boundary contact forces are calculated, refer to the DEM Technical Manual. (From the Rocky program **Help** menu, point to **Manuals**, and then click **DEM Technical Manual**.)

8.2.7. I do not know how to simulate magnetic forces

It is quite possible to simulate different types of magnetic forces in Rocky by adjusting the **Adhesive Force** (see also About Physics Parameters (p. 417)) and various material-to-material properties, including the **Adhesive Distance** of your particle-to-particle and/or particle-to-boundary interactions.

See Also:

- About Modifying Materials Interactions and Adhesion Values (p. 597)

8.2.8. I need help simulating an HPGR

High Pressure Grinding Rolls (HPGRs) are comminution devices that are simulated in Rocky using breakage modeling. Use the following resources to help you get started simulating an HPGR device:

- See a video example of an HPGR simulation here: Rocky DEM - Showing Particle Breakage in HPGR Mill
- Walk step-by-step through setting up an processing an HPGR simulation here:

Tutorial - High Pressure Grinding Roll (HPGR) in the *Rocky Tutorial Guide*

See Also:

- Enable Particle Breakage Calculations (p. 397)

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

8.2.9. I cannot find the upper limit for importing custom particles shapes

Each custom particle shape that you import counts as one separate Particle set (p. 1393), and there is no limit to the number of unique Particle sets you can add per simulation project.

See Also:

- About Defining and Importing Custom Particle Shapes (p. 670)
- About Adding and Editing Particle Sets (p. 602)

8.2.10. I entered an invalid variable or value and Rocky accepted it

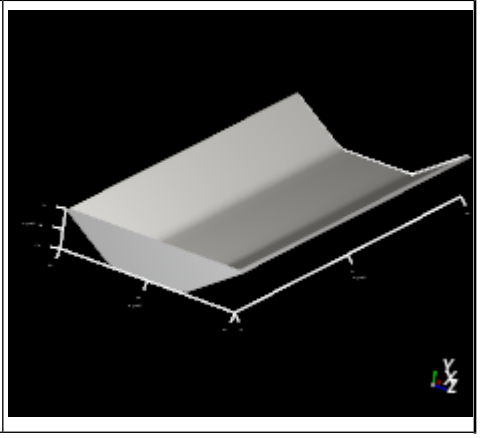
Most Rocky fields enable you to enter variables and/or mathematical functions instead of direct, numerical values. (See also Enter Input Variables or Mathematical Functions as Parameter Values (p. 372).)

Because this parametric variables ability enables you to change the value of any field making use of an input variable at any point during your project setup, Rocky is not able to pre-limit the values you enter to only accept valid entries. Rather, Rocky notifies you of invalid entries afterward through either a dialog or **Status** panel warning (see also Double-Click the Status Panel to Jump to the Appropriate UI Location (p. 117)).

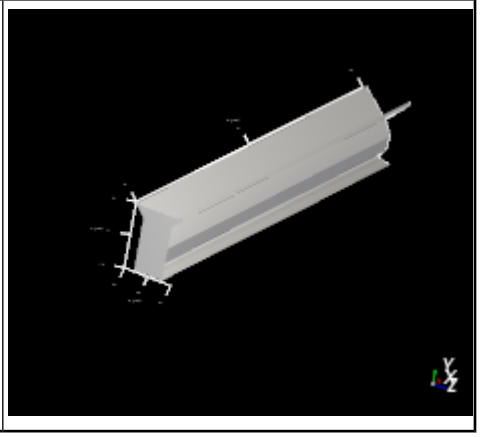
Therefore, it is best to follow the **Range** column provided for the parameter's definition in Rocky Help (this document) and heed the warnings in the **Status** panel rather than assuming that if Rocky accepts the value, it is valid for your simulation setup.

For example, Rocky will allow you to set the **Troughing Angle** parameter for a receiving conveyor (with a **Belt Profile** equal to two or more rolls) to any value even though some values make less sense than others, and others are simply invalid, as illustrated in Table 1.

Table 1: Example results when different Troughing Angles are provided for receiving conveyors with two or more belt profiles

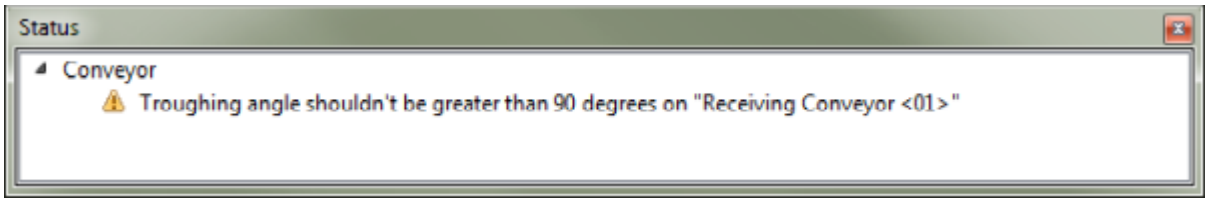
Troughing Angle Value	Results Description	Results Image
35 degrees (default value)	Valid entry produces a troughed belt that will hold material.	

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

Troughing Angle Value	Results Description	Results Image
0 (zero) degrees	Invalid entry produces no belt.	
135 degrees	Valid but useless entry produces a folded over belt that will hold no material.	

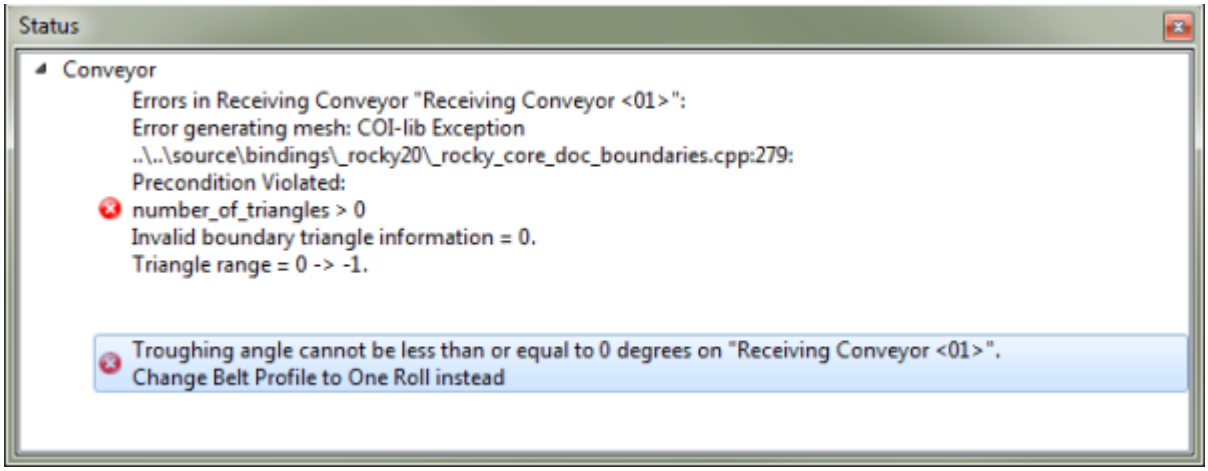
Rocky Help defines the **Range** for this **Troughing Angle** parameter as "0 < 90 degrees", which can help guide you before you enter your values. And if you do enter a value outside of this range, the following **Status** panel warnings are presented:

Figure 8.3: Warning provided in Status panel when Troughing Angle value is above the recommended maximum



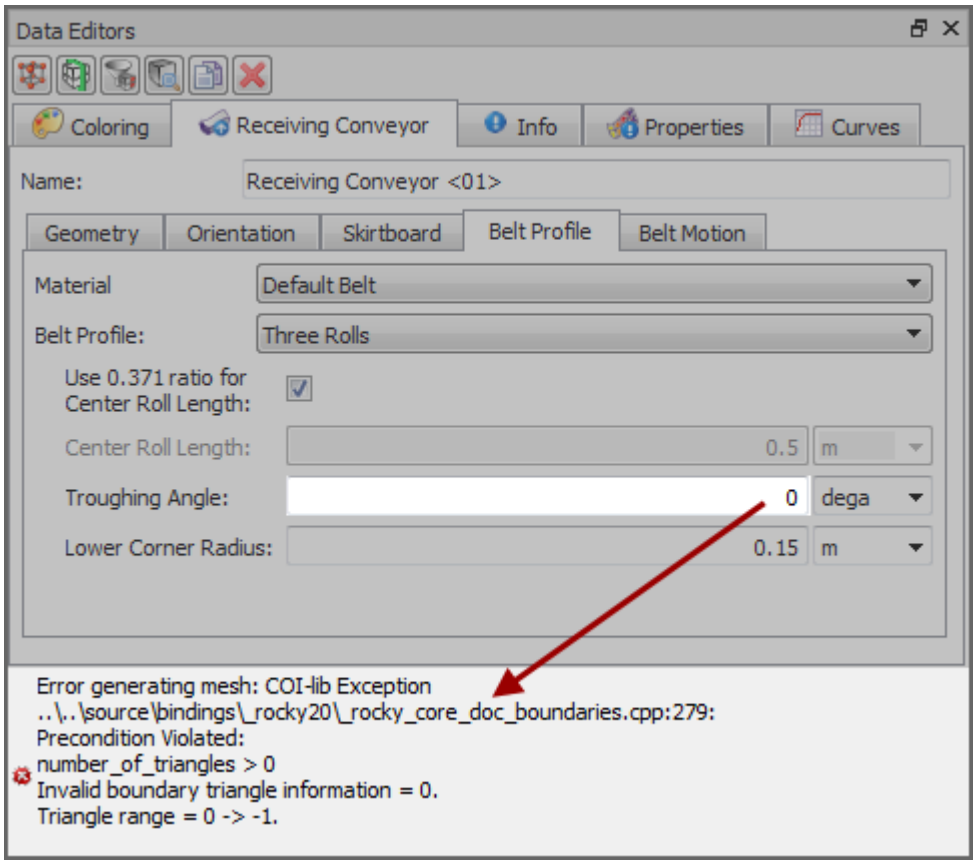
Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

Figure 8.4: Warning provided in Status panel when Troughing Angle is below the recommended minimum



In addition, when a 3D View window is open, some errors are also presented at the bottom of the Data Editors panel:

Figure 8.5: Error shown on Data Editors panel when zero is entered for Troughing Angle



See Also:

- Troubleshooting (p. 1315)
- Enter Input Variables or Mathematical Functions as Parameter Values (p. 372)

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

- About Receiving Conveyor Parameters
- About Feed Conveyor Parameters

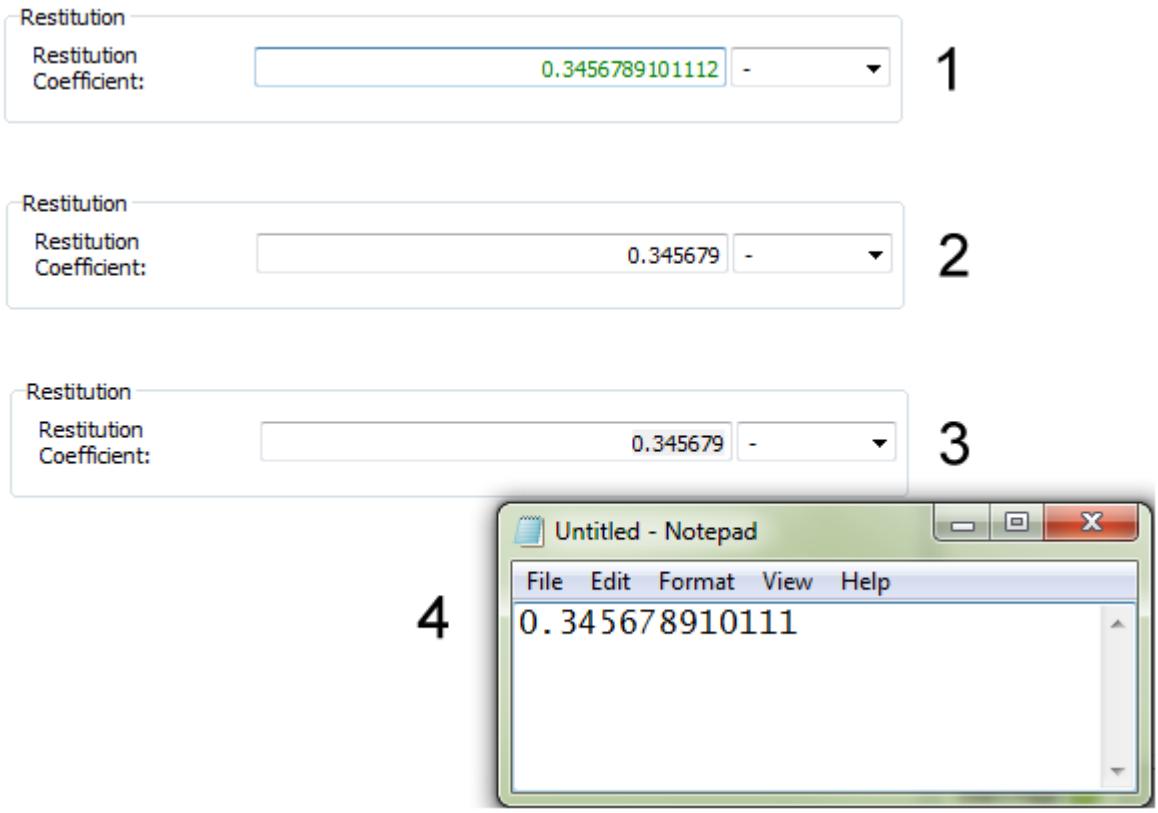
8.2.11. I briefly see long numbers when pasting in copied values

When displaying calculated values in text fields, Rocky keeps long, very precise numbers as an internal value but will truncate what is displayed in the UI to only six decimal places. When you copy (Ctrl + C) a value like this, the full internal value is the one copied to your clipboard and not the truncated version.

This means that if you then paste the copied value into another program, you will see the full, untruncated value. However, if you paste it into another field in Rocky, you will briefly see the full, untruncated value but then when you click away from that field, Rocky will truncate it to six decimal places again.

In the example shown in Figure 1, a very precise value is entered with 13 numbers after the decimal place (1). After clicking away from the text field, that precise value is truncated to only 6 numbers after the decimal place (2). However, if you select and copy that truncated value (3), and then paste it into a program outside of Rocky (4), you get the full internal value with all the decimal places.

Figure 8.6: Example of a long number (1) truncated (2), copied (3), and then pasted outside of Rocky (4)



See Also:

- Use Colored Text to Validate the Syntax of Your Entries (p. 117)

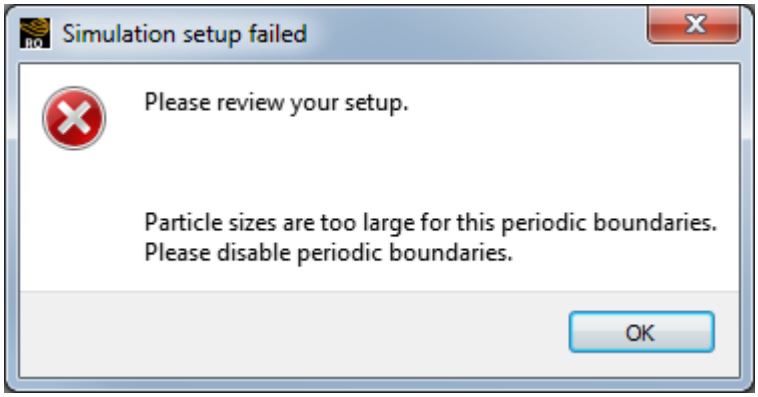
Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

8.2.12. My particle size is too large for my periodic domain

When setting periodic Cartesian domains (previously known as "periodic boundaries"), it is important to set the distance between the **Min Coordinate** and **Max Coordinate** values to at least 2.5 times wider than your largest particle size. Or, alternatively, ensure that your largest particle size is less than 0.4 times the width of your periodic Cartesian domain.

If you set the width of the periodic domain smaller than this, you will see the following error message after starting to process your simulation.

Figure 8.7: Simulation setup failed message



To solve this issue, click **OK** and then change your periodic Cartesian domain's **Min Coordinate** and **Max Coordinate** values, and/or your particle sizes, to be inline with the guidelines above.

See Also:

- About Domain Settings Parameters (p. 452)
- About Adding and Editing Particle Sets (p. 602)
- Particles are going through the walls of my geometry (p. 1353)

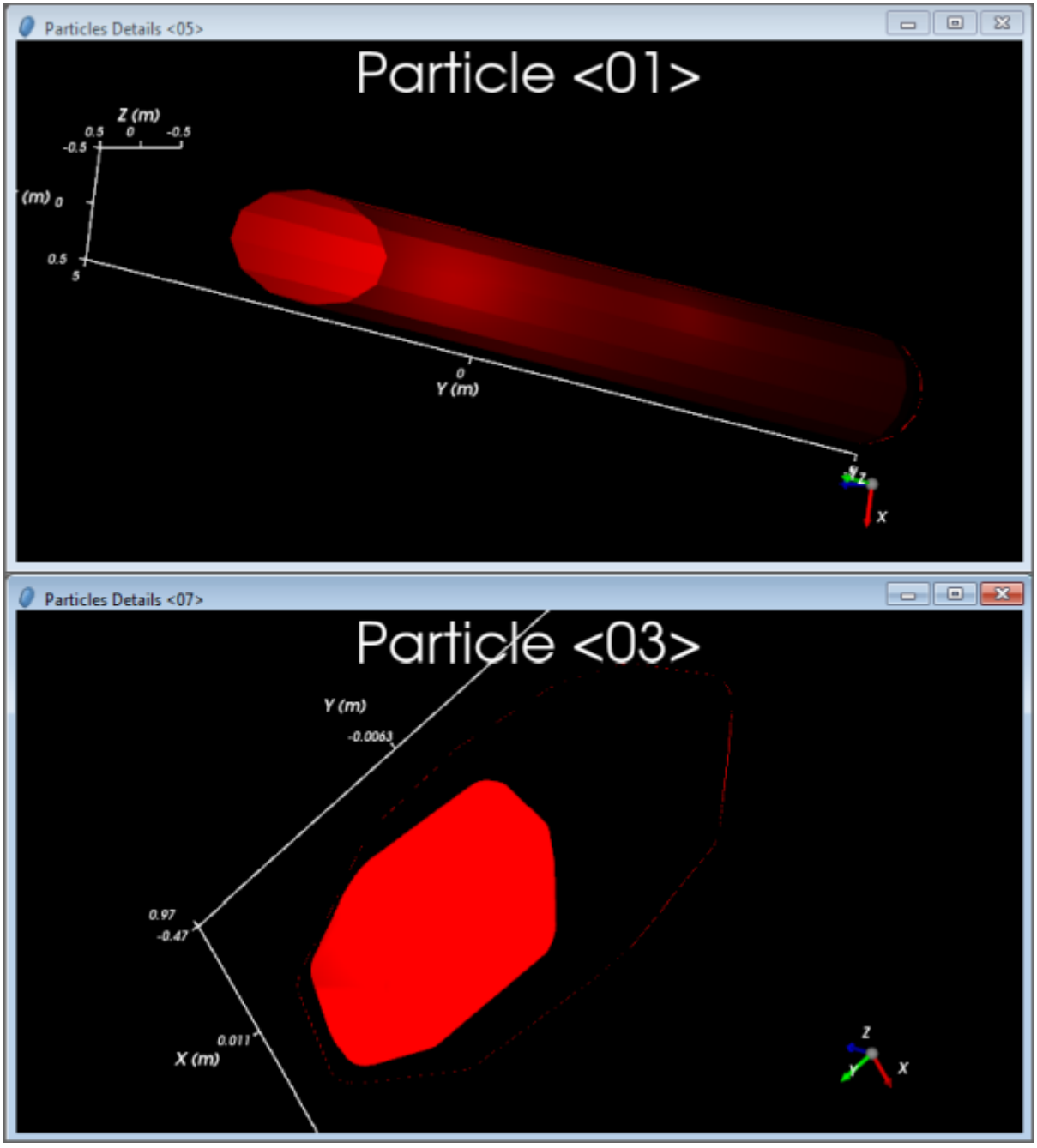
8.2.13. Parts of my particle disappear or appear cut off

In Rocky, rendering and lighting-both of which are used to display 3D items in view windows-do not always work well with items that are very small in scale. This is not usually an issue with 3D Views because the scale of your geometries is typically large enough to prevent these kind of issues.

It can be an issue, however, when previewing very small particle shapes in a Particles Details window. This is because the display of the shape is linked to the size of the particle. Especially when going from a (relatively) large particle size display to a very small one in the same window, zooming in on the resulting much smaller particle may result in a display that cuts off part of the particle, and/or makes certain particle areas transparent in unusual ways (Figure 1).

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

Figure 8.8: Example of a small-scale straight fiber (above) and a small-scale sphero-polyhedron (below) displaying rendering and lighting issues in the Particles Details window



If this happens, reset the view by doing one of the following:

- Press the **R** key on your keyboard or press the **Fit** button on the toolbar to fit the active components into view
- Close the current Particles Details window and open a new one

See Also:

- About Adding and Editing Particle Sets (p. 602)

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

- About Particles Details Windows (p. 679)

8.2.14. Rocky does not list my Ansys version when I try to set up 2-Way Fluent Coupling

In order to list the Ansys versions available for 2-Way Fluent coupling, Rocky needs information about the Ansys installation. Rocky follows a particular order when making that determination, and that order differs slightly depending upon whether your installation is on a Linux- or Windows-based machine. See below for further details and possible solutions.

Order Rocky Follows When Looking for Ansys Versions

1. On both Windows- and Linux-based machines, Rocky first expects a variable to be provided called `AWP_ROOTXXX`, where XXX refers to the particular Ansys version. Rocky expects this variable to point to the root folder of the AWP folder. For example:
   - On Windows-based machines, this variable for Ansys version 19.0 could point here: `C:Program FilesANSYSv190` Tip:On Windows machines, this variable should have been added during the Ansys installation process.
   - On Linux-based machines, this variable for Ansys version 19.0 installed for "user" could point here: `/home/user/ansys/v190`
2. If the `AWP_ROOTXXX` variable is not provided as expected, then the following steps are taken:
   - On Windows-based machines, Rocky does nothing further.
   - On Linux-based machines, Rocky takes the following steps in the order provided:
     a. Rocky expects a different variable to be provided called `ANSYSXXX_DIR`, where XXX refers to the particular Ansys version. Rocky expects this variable to point to the ansys folder inside of the AWP folder. For example: `/home/user/ansys/v190/ansys`
     b. If the `ANSYSXXX_DIR` variable is not provided as expected, then Rocky looks for the Ansys installation files in the locations and order listed:
        – Inside the version folder of the `ansys_inc` folder. For example, `/ansys_inc/v190`
        – Same folder as before but inside the `Usr` folder. For example, `/usr/ansys_inc/v190`

Possible Solutions

If when you are setting up a 2-Way Fluent coupling simulation, Rocky does not list the Ansys version you have installed on your machine, start by checking if any of the options above exist (paths or variables).

If you find they do not exist, then Rocky does not know where Fluent is installed and cannot recognize whether it is valid for 2-Way Fluent coupling. You can try solving this by creating an environmental variable as suggested above, or by reinstalling Ansys on your machine.

See Also:

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

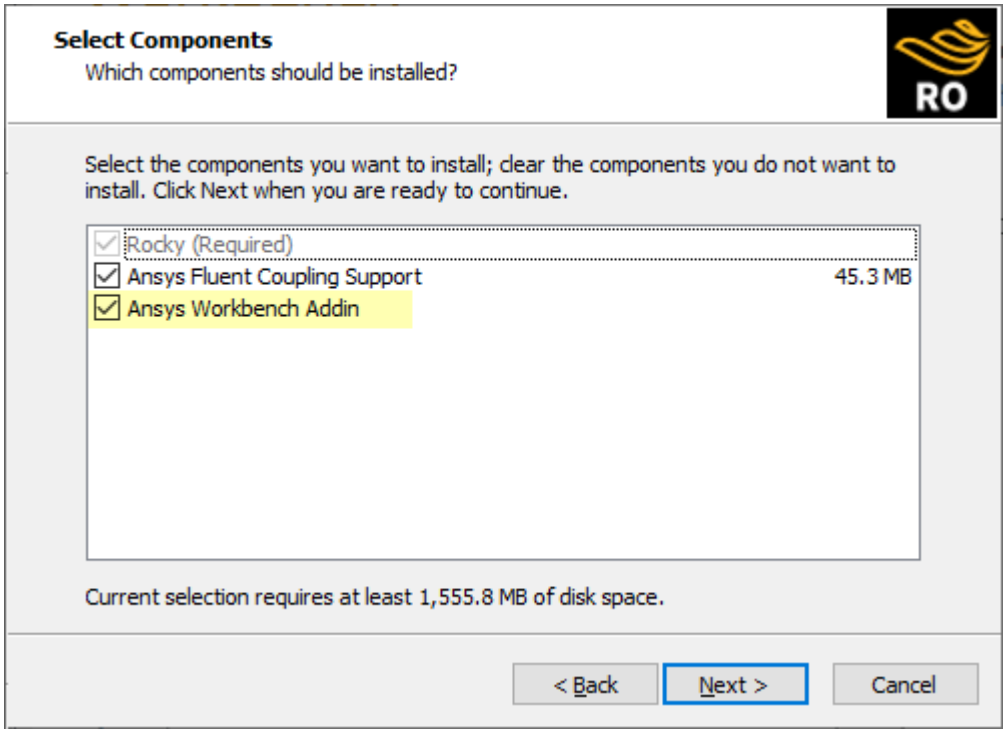
- Use the 2-Way Fluent Method to Process Fluent and Rocky Simulations (p. 386)
- About Using the 2-Way Fluent Method (p. 795)

8.2.15. I cannot find the Rocky component in Workbench

If when you open Ansys Workbench, you do not see the **Particle Dynamics (Rocky)** component listed under **Analysis Systems** in the **Toolbox** panel, it is likely that you do not have the Ansys Workbench Addin installed. In this version, the only way to install this addin is to re-install Rocky.

During the re-installation process, ensure that you keep the **Ansys Workbench Addin** checkbox selected (Figure 1).

Figure 8.9: Rocky install screen showing Ansys Workbench Addin selected



See Also:

- About Rocky and Ansys Workbench Integration (p. 144)
- Install Ansys Coupling Components (p. 792)

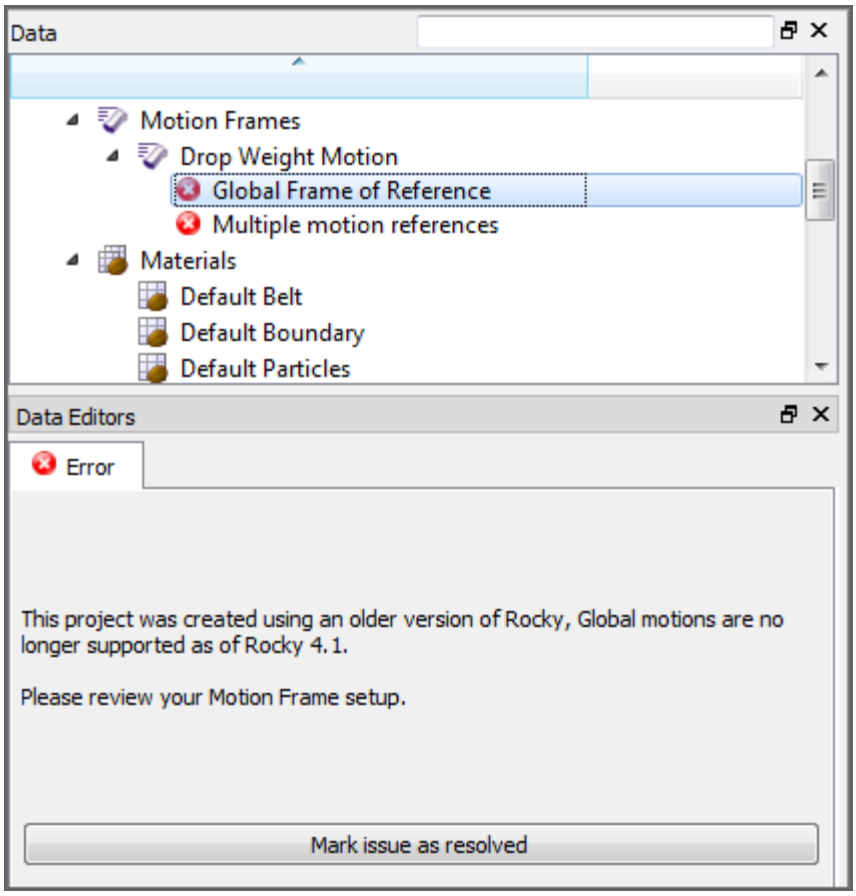
8.2.16. I get "reference" errors in my Motion Frames when I open an older project

In versions of Rocky prior to v4.4, there was a **Reference** setting that allowed one of several values (Parent, Local, or Global, depending upon the Rocky version) to be set on an individual motion or Frame (depending upon the Rocky version). In this version of Rocky, the **Reference** setting has been removed. This means that all motions within a Frame are always based upon what was previously known as the "local" reference.

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

Because of these changes, if you open a project created prior to Rocky v4.4 that had either different motion-based References set for one Frame, or had either **Global** or **Parent** References set for either frames or individual motions, you will receive special errors in the Data panel (Figure 1).

Figure 8.10: Two different Motion Frame references errors appearing in the Data panel



These errors are meant to communicate that the older project you opened has reference values set that are no longer supported in this newer version, and Rocky needs you to verify that the new Frame-based local-only reference settings are as you want them before continuing.

To solve these errors:

1. From the **Data** panel under **Motion Frames**, select the Frame over which the error appears.

2. From the **Data Editors** panel, ensure that the motion settings are as you want. Important As you review the settings, keep in mind that all motions within the Frame are now based only on an implicit "local" reference. Tip:You can recreate similar "parent" reference behavior by using nested Frames. See also I cannot figure out how to set the Parent Reference for a Motion Frame (p. 1332).

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

3. Once you have ensured that the settings are as you want, from the **Data** panel, select the error and then from the **Data Editors** panel, click the **Mark issue as resolved** button. The error is removed.

Note:

You must address these errors before you can process your simulation.

See Also:

- About Creating and Applying Motion Frames (p. 533)

8.2.17. I cannot find the Forces for FEM Analysis checkbox

The **Collect Forces to FEM Analysis** checkbox used to be on the **Solver** | **General** tab but with the inclusion of the new Boundary Collision Statistics feature, is now enabled in that location via the **Forces for FEM Analysis** checkbox.

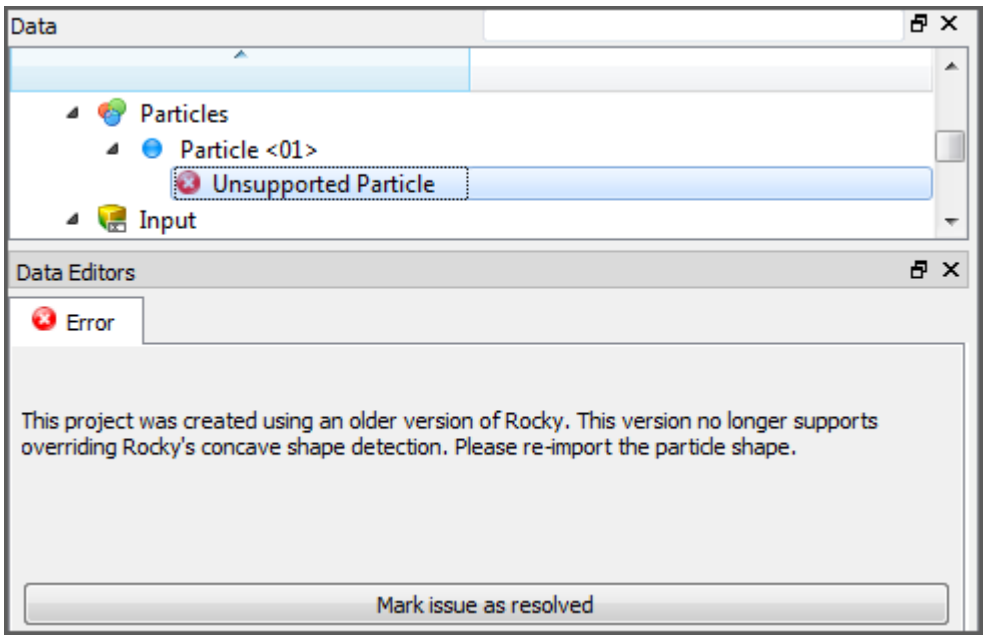
See Also:

- About Collision Statistics for Boundaries (p. 312)

8.2.18. I get an "Unsupported Particle" error when I open an older project

In previous versions of Rocky, you were allowed to mark a particle shape as convex even though Rocky determined it to be concave. Overriding this determination is no longer supported in newer versions of Rocky. Therefore, when you open an older project where the concavity of the particle shape was overridden, you will see an "Unsupported Particle" error associated with the Particle set (p. 1393) (Figure 1).

Figure 8.11: Example of a particle set with an Unsupported Particle error



To solve this issue, you will need to re-import the shape, and then mark the issue as resolved by clicking the **Mark issue as resolved** button.

If after re-importation, Rocky determines the shape to be concave but you still want Rocky to calculate it as convex, you will be given the option of having Rocky re-draw the particle to meet the latest shape requirements. (See also About Defining and Importing Custom Particle Shapes (p. 670).)

See Also:

- About Defining and Importing Custom Particle Shapes (p. 670)
- About Adding and Editing Particle Sets (p. 602)

8.2.19. I cannot find Particle Collision Statistics

In prior versions of Rocky, there was a feature on the **Solver** | **Collisions** tab known as *Particle Collision Statistics*. In this version of Rocky, that feature is now known as *Intra-particle Collision Statistics* and is now enabled from a **Module** with that same name.

See Also:

- About Intra-Particle Collision Statistics (p. 324)
- About the Intra-particle Collision Statistics Module (p. 880)

8.2.20. I cannot find Particle Size Scale Factor

In previous versions of Rocky, there was an Experimental (Beta) Feature for CFD Coupling called **Particle Size Scale Factor**. In this version of Rocky, that setting has been removed and replaced with a new Coarse Grain Modeling setting called **CGM Scale Factor**. This new parameter can be found on the **Size** tab for a Particle set when Coarse Grain Modeling is enabled. (See also About Adding and Editing Particle Sets (p. 602).)

Note:

Older Rocky simulations that made use of the prior **Particle Size Scale Factor** may not work the same in this version of Rocky. It is recommended that for these older simulations, you enable Coarse Grain Modeling, make use of the new **CGM Scale Factor** setting, and then re-process your simulation.

See Also

- About Adding and Editing Particle Sets (p. 602)
- About Setting Global Preferences (p. 227)
- About Physics Parameters (p. 417)

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

8.2.21. I cannot find the Leeds Adhesion Force model

In previous versions of Rocky, there was an **Adhesive Force** (on the **Physics** | **Momentum** tab) called **Leeds**.

As of the Rocky v4.4 release, the Leeds Adhesion model has been removed from the Rocky product and converted into an external module (p. 1391), which is installed separately via a ZIP file.

To make use of the Leeds Adhesion model, you must gain access to and then install this external module. Specifically:

- To gain access to the module, visit the Ansys Download Center page, navigate to "Current Release" and find the "AddOn Packages" tab. From the Ansys Rocky box, download the SDK and Modules Full Package. Once the download concludes, you can find the **Leeds contact model** module in the Ready-to-use Modules folder.
- To install the module once you have access to it, refer to the Install an External Module (p. 223) topic.

See Also:

- About Physics Parameters (p. 417)
- About Modules Parameters (p. 431)

8.2.22. I cannot find the Inlet Start and Stop Times

In previous versions of Rocky, you could set an Inlet's Start and Stop Time, as well as define periodic release criteria on a separate "Simulation Configuration" sub-tab for the Inlet.

As of Rocky v4.4, these same "Simulation Configuration" parameters have been moved to the Continuous Injection Input. This means that you will now set these parameters per **Entry Point** that you define, which includes both Inlets and any default Feed Conveyors you have added to your simulation.

See Also:

- About Inlet Parameters
- About Feed Conveyor Parameters
- About Adding and Editing Particle Inputs (p. 708)

8.2.23. I cannot find the Movements Tab for my older simulation projects

Simulation projects created in versions of the software prior to Rocky 4 were able to define custom (imported) geometry motions by using a **Movements** tab in the **Data Editors** panel. Opening these older simulation projects in Rocky versions 4.0.0 - 4.3.x would retain the ability to still use these older movements even though the **Motion Frames** feature was created to replace this older functionality.

As of Rocky v4.4, these older movements are no longer supported. If you happen to open an older simulation project that had movements defined, you will now need to re-define those motions using the **Motion Frames** feature.

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

See Also:

- About Creating and Applying Motion Frames to Custom Geometries (p. 533)

8.2.24. I cannot figure out how to set the Parent Reference for a Motion Frame

In previous versions of Rocky, it was possible to set the reference for a Motion Frame to either "local" or "parent". When a parent reference was selected, the next motion for the Frame was based not upon the current orientation of the Frame, but upon the original orientation of the Frame when the motion started, relative to the current position of its parent Frame (if it was a child Frame) or the Global Coordinate System (if the Frame had no parent).

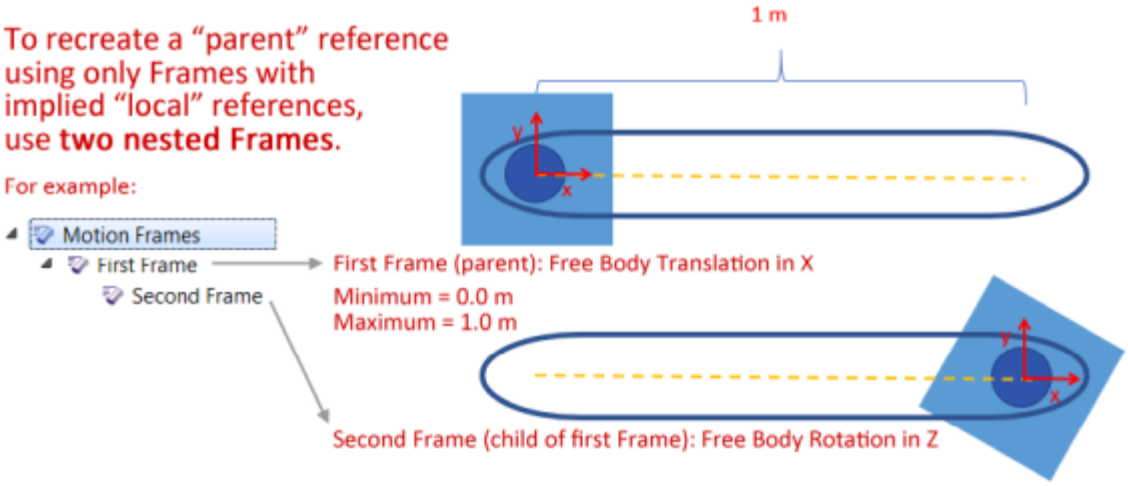
In this version of Rocky, all Frames are now implicitly based only upon what was once known as the "local" reference, which uses the current orientation of the selected Frame to define the next movement. In this way, the reference is always moving along with the Frame.

Important When opening a project created in an older version of Rocky, any Frames with references set to "parent" will now act only as "local". See also I get "reference" errors in my Motion Frames when I open an older project (p. 1327).

In this version of Rocky, you can recreate this "parent" reference behavior by creating a nested (child) Frame underneath another Frame (parent).

For example, in Figure 1 the First Frame (parent) contains a translation motion, and the Second Frame nested beneath it (child) contains a rotation motion. When the child Frame is assigned to a geometry, the rotation (child) motion will move relative to the current position of the translation (parent) motion.

Figure 8.12: Example showing how nested Frames can recreate a "parent" reference



See Also:

- About Creating and Applying Motion Frames to Custom Geometries (p. 533)

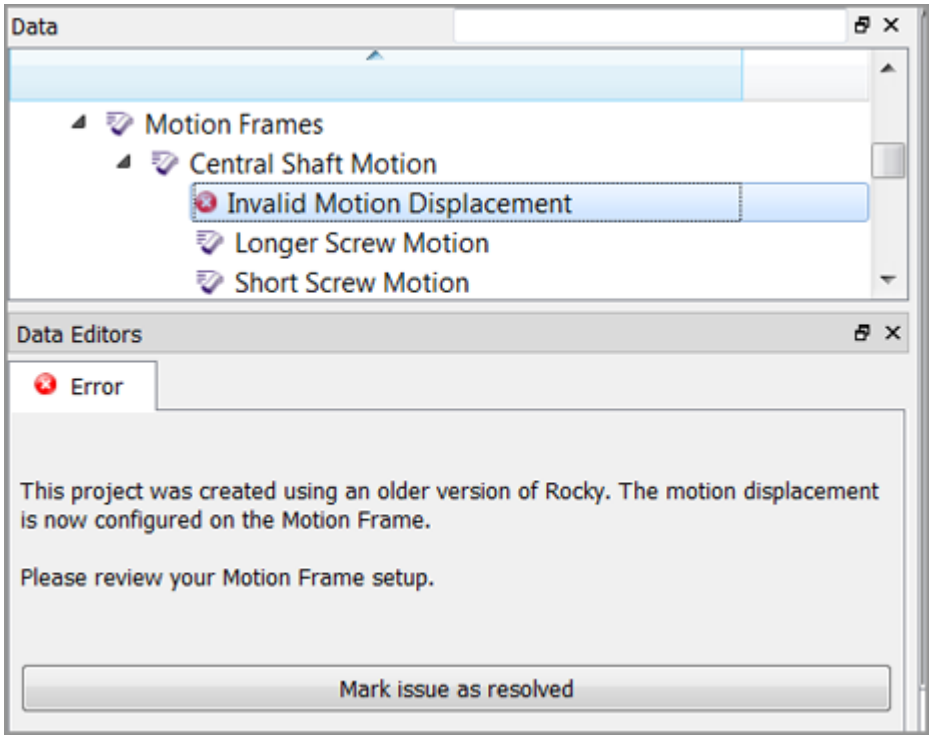
Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

8.2.25. I get "Displacement" errors in my Motion Frames when I open an older project

In versions of Rocky prior to v4.4, you were able to set "Keep in place" or "Displace with movement" options for certain motions within a Motion Frame. This meant that it used to be possible to have a Frame with both motions that were kept in place and others that displaced. In this version of Rocky, displacement is set only once for the entire Frame.

Because of these changes, if you open an older project containing a Frame with individual motions that had both "keep in place" and "displace with motion" options set, you will receive a special error in the Data panel (Figure 1).

Figure 8.13: A Motion Frame displacement error appearing in the Data panel



This error is meant to communicate that the older project you opened had mixed displacement settings that are no longer supported in this newer version, and Rocky needs you to verify that the new Frame-based displacement settings are as you want them before continuing.

To solve these errors:

1. From the **Data** panel under **Motion Frames**, select the Frame over which the error appears.
2. From the **Data Editors** panel, ensure that the motion settings are as you want. Important As you review the settings, keep in mind that all motions within the Frame will now share only one kind of displacement option.

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

3. Once you have ensured that the settings are as you want, from the **Data** panel, select the error and then from the **Data Editors** panel, click the **Mark issue as resolved** button. The error is removed.

Note:

You must address these errors before you can process your simulation.

See Also:

- About Creating and Applying Motion Frames to Custom Geometries (p. 533)

8.2.26. My "Air Viscosity" units changed

If you are using the **1-Way LBM** CFD Coupling method, you may have noticed that the parameter known in versions prior to v4.5.0 as **Air Viscosity** is now known as **Air Kinematic Viscosity**. In addition, the default units have changed from Pascal seconds (Pa·s) to square meter per seconds (m:sup:2/s) even though the default value has stayed the same.

While the default value of 1.5e-05 was correct for kinemetic viscosity, which is how Rocky actually uses this parameter, the units were incorrectly listed, which gave the false impression that this parameter represented dynamic viscosity.

Therefore, if based on the incorrect units, you misunderstood this parameter to be dynamic viscosity, you might have changed the default value for air viscosity to 1.8e-05 Pa·s. This value, while correct for dynamic viscosity, is not correct for kinemetic viscosity, which is how Rocky actually interprets it.

Since the numerical values for air are very close by both measures of viscosity (1.8e-05 Pa·s vs. 1.5e-05 m^2/s at 20 °C), this error is unlikely to have affected your simulations. But in cases where exact values of air viscosity are required, or in cases where you are comparing LBM results from different Rocky versions, please be aware that adjustments might need to account for this change.

See Also:

- About Using the 1-Way LBM Method (p. 771)
- About Setting Your Unit System (p. 238)

8.2.27. I cannot enter an input variable or mathematical function into a text field

Most text fields in Rocky support entering variables or mathematical functions as parameter variables, including three-part text fields. (See also Enter Input Variables or Mathematical Functions as Parameter Values (p. 372).)

There are several text fields where parametric variables are currently not supported, however. These are listed in the table below.

Table 1: Text fields for which entering variables or mathematical functions are not supported

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

Text Field for which Parametric Variables are Disabled	Where Located	See Also
Size	Under **Custom Size** on the **Particle** | **Info** tab for an individual Particle set.	About Adding and Editing Particle Sets (p. 602)
Maximum Iterations	When the **Advanced Features** checkbox is enabled, this is located on the **2-Way Fluent** | **Coupling** tab for 2-Way Fluent CFD Coupling.	About Using the 2-Way Fluent Method (p. 795)
Solver Processes	On the **2-Way Fluent** | **Fluent** tab for 2-Way Fluent CFD Coupling.	About Using the 2-Way Fluent Method (p. 795)
Files to keep	On the **2-Way Fluent** | **Fluent** tab for 2-Way Fluent CFD Coupling.	About Using the 2-Way Fluent Method (p. 795)
Contact Stiffness Multiplier	When the **Advanced Features** checkbox is enabled, this is located under **Momentum** on the **Materials Interactions** tab for a material-material pair.	About Modifying Materials Interactions and Adhesion Values (p. 597)
Close Packing Volume Fraction	Under **Coarse Grain Modeling** on the **Physics** | **Coarse-Graining** tab.	About Physics Parameters (p. 417)
Solver Curves	Under **Output Frequencies** on the **Solver** | **Time** tab.	About Solver Parameters (p. 464)
Rendering Scale	On the **Particles** tab for the **Windows Editors** panel when the main 3D View entity is selected.	About Using the Window Editors Panel to Change All 3D Views in the Project (p. 1048)

See Also:

- Enter Input Variables or Mathematical Functions as Parameter Values (p. 372)
- Defining Your Variables (p. 243)

8.2.28. I cannot find the Velocity Dependent Adhesive Force model

In previous versions of Rocky, if you enabled the **Experimental (Beta) Features** checkbox on the **Options** | **Preferences** dialog (see also About Setting Global Preferences (p. 227)), there was an **Adhesive Force** model (on the **Physics** | **Momentum** tab) called **Velocity Dependent**.

As of the Rocky v4.5 release, the Velocity Dependent Adhesion model has been removed from the Rocky product. If you want to simulate an older project that made use of this model, you must select a different Adhesive Force model before you will be able to process it.

See Also:

- About Setting Global Preferences (p. 227)

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

- About Physics Parameters (p. 417)

8.2.29. I cannot find the Torque Law setting for 1-Way Constant

In prior versions of Rocky, you could define a **Torque Law** for your 1-Way Constant particle-fluid interactions. (See also About Using the 1-Way Constant Method (p. 763).) However, the 1-Way Constant calculations did not actually make use of the **Torque Law** setting, so as of Rocky v4.5.0, this setting has been removed.

You are still able to define a **Torque Law** for 1-Way Fluent simulations. (See also About Using the 1-Way Fluent Method (p. 775).)

See Also:

- About Using the 1-Way Constant Method (p. 763)
- About Using the 1-Way Fluent Method (p. 775)

8.2.30. I cannot find the Output Frequency parameter

In previous versions of Rocky, there was a parameter on the **Solver** | **Time** tab called **Output Frequency**. In this version of Rocky, that parameter is now called **Simulation** and is located under the **Output Frequencies** section.

See Also:

- About Solver Parameters (p. 464)
- Troubleshooting Simulation Setup Problems (p. 1338)

8.2.31. I cannot find the Loading N-Steps parameter

In previous versions of Rocky, there was a parameter on the **Solver** | **Time** tab called **Loading N-Steps**. In this version of Rocky, that parameter has been re-classified as an advanced parameter, and is now located on the **Solver** | **Advanced** tab.

Note:

The **Advanced** tab only appears when the **Advanced Features** checkbox is enabled on the **Additional Features** tab of the **Options | Preferences** dialog. (See also About Setting Global Preferences (p. 227).)

See Also:

- About Solver Parameters (p. 464)
- About Setting Global Preferences (p. 227)
- Troubleshooting Simulation Setup Problems (p. 1338)

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

8.2.32. I cannot find the Set Contact Detection Neighboring Distance parameter

In previous versions of Rocky, there was a parameter on the **Solver** | **Advanced** tab called **Set Contact Detection Neighboring Distance**. This parameter enabled you to set only a single **Neighboring Distance** value for both particle-to-particle and particle-to-boundary contacts.

In this version of Rocky, you can now set unique **Neighboring Distance** values for each type of contact:

- For particle-to-particle contacts, this new parameter is called **Set Contact Detection Neighboring Distance between Particles**.
- For particle-to-boundary contacts, this new parameter is called **Set Contact Detection Neighboring Distance between Particles and Triangles**.

Note:

The **Advanced** tab only appears when the **Advanced Features** checkbox is enabled on the **Additional Features** tab of the **Options | Preferences** dialog. (See also About Setting Global Preferences (p. 227).)

See Also:

- About Solver Parameters (p. 464)
- About Setting Global Preferences (p. 227)
- Troubleshooting Simulation Setup Problems (p. 1338)

8.2.33. I cannot find the Export Solution to Rocky item in Fluent

In previous versions of Rocky, if you had installed the **Ansys Fluent Coupling Support** item (see also Install Ansys Coupling Components (p. 792)), you could choose to export data out of Fluent through an **Export Solution to Rocky** menu item that was located within Fluent.

In this version of Rocky, this menu item has been replaced with another, more expansive **Rocky Export** menu in Fluent. Refer to the About Using the 1-Way Fluent Method (p. 775) topic for more information.

See Also:

- Troubleshooting Simulation Setup Problems (p. 1315)
- About Using the 1-Way Fluent Method (p. 775)
- Install Ansys Coupling Components (p. 792)

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

8.2.34. I cannot find the Maximum Volume Fraction Target parameter for 2-Way Fluent

In prior versions of Rocky, when the **Advanced Features** setting was enabled, you could define the **Maximum Volume Fraction Target** when you selected **Volumetric Diffusion** as your **Mapping Method** for your 2-Way Fluent simulations. (See also About Using the 2-Way Fluent Method (p. 795).) However, this parameter was renamed to **Solids Maximum Volume Fraction Target** to help clarify that this setting pertains to the solids (particles) only, and not to the fluids.

See Also:

- About Using the 2-Way Fluent Method (p. 795)

8.3. Troubleshooting Simulation Processing Problems

What kind of problem are you having?

- Troubleshooting Common Processing Errors (p. 1339)
- Other Simulation Processing Issues:
  - I am processing my simulation but no output files are being created (p. 1352)
  - I get unusual results when I process my simulation (p. 1351)
  - I set parameters for my simulation but I cannot process it (p. 1351)
  - My particles are flowing too quickly (p. 1353)
  - Particles are going through the walls of my geometry (p. 1353)
  - My simulation is taking too long to process (p. 1351)
  - No particles appear when I process the simulation (p. 1355)
  - Particles stopped flowing before my simulation was done processing (p. 1356)
  - I cannot find a batch mode for processing (p. 1356)
  - I want to use a Pascal-based GPU card instead of the recommended CUDA-based one (p. 1356)
  - I cannot find where to set the number of cores for CPU processing (p. 1357)
  - A fiber or other particle shape disappeared from my simulation during processing (p. 1357)
  - My GPU card is not being recognized as CUDA compatible (p. 1357)
  - I cannot resume processing my older simulation in the latest version of Rocky (p. 1359)
  - My inlet stopped injecting particles for a long time and then later resumed it (p. 1359)
  - My Volume Fill isn't working right (p. 1360)

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

- I cannot change my setup parameters during processing (p. 1361)
- My particle's height increases after each collision (p. 1362)

See Also:

- Troubleshooting (p. 1315)

8.3.1. Troubleshooting Common Processing Errors

What kind of problem are you having?

- I get a "runtime" or "system resource" error when I try processing my simulation (p. 1339)
- I get a "CPU" or "GPU" error message when I start processing my simulation (p. 1341)
- I get a "Simulation setup failed" message when I start processing my simulation (p. 1345)
- I get a "No Solver license available" message when I run the Rocky Scheduler but the status says "License Ok" (p. 1340)
- I get an "Error simulating" or "Simulation error" message when I resume or start processing my simulation (p. 1342)
- I got a message saying concave particles are not allowed in my Rocky version (p. 1341)
- I get a "Failed to write the log" message while processing my simulation (p. 1340)
- I get warnings or errors on my Simulation Log Panel (p. 1346)
- I get an "Error copying memory from device to host" message when I try processing my simulation (p. 1348)
- I get a "Particles composed of Multiple Elements can't have size defined" error when I try processing my simulation (p. 1349)
- I get a scripting error when I attempt to run the simulation in batch mode (p. 1349)
- I get a "Server returned a fault exception" error when running a script in batch mode (p. 1350)

See Also:

- Troubleshooting Simulation Processing Problems (p. 1338)

I get a "runtime" or "system resource" error when I try processing my simulation

If you get a "runtime" or "system resource" error when you try processing your simulation, then your system probably cannot handle the amount of data you are trying to process.

See My simulation is taking too long to process (p. 1351) for suggestions.

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

I get a "No Solver license available" message when I run the Rocky Scheduler but the status says "License Ok"

Most users of Rocky will have single-instance licenses, which means that only one instance of the Rocky solver can be running at any one time. For these users, it is possible to use the Rocky Scheduler program to run multiple simulations in series, or one-by-one, but not in parallel or simultaneously.

If you have a single-instance license and attempt to start processing more than one simulation at a time, you will get a "No Solver license available" message for all but the first simulation, as it will be using the single license that was available. Once that first simulation is done processing, the license will be freed up to be used by the next simulation in the queue, and so on.

If you have a need to process multiple simulations simultaneously, please Submit a Support Request (p. 1293) or contact your Rocky representative to ask about upgrading to an unnumbered Rocky license.

See Also:

- About Processing Multiple Simulations In Succession (p. 920)

I get a "Failed to write the log" message while processing my simulation

If while processing your simulation, you see a "Failed to write the log" message on your Rocky screen or in the Rocky Scheduler Progress column, this likely means that Rocky has stopped processing your simulation for one of the reasons explained below.

Important Because of the reasons below, when choosing to processes very long or complex cases- especially when over a network -it is important to periodically check the status of your simulation to ensure that loss of connectivity or other issues haven't stopped it from processing.

CONNECTIVITY WAS LOST WHEN PROCESSING OVER A NETWORK

This version of Rocky requires continuous access to the .rocky20.log file in order to process your simulation. This requirement means that when processing over a network, any loss of connectivity between the system running Rocky and the network location storing your Rocky files will result in an automatic stop of simulation processing.

This is done to avoid the situation where Rocky continues to process your simulation but is unable to save your data. Preventing the simulation from processing further helps avoid any losses or breaks in your simulation data collection. This allows you to simply resume processing your simulation once connectivity is reestablished, rather than having to start processing over from the beginning.

To resolve this, check your network connection, and then resume processing your simulation again.

THE TARGET DRIVE BECAME TOO FULL

If the drive to which you are saving your Rocky files becomes too full to add more files, you might also see this error.

To resolve this, do one of the following:

- Clear your current drive of other files (unrelated to your current simulation) to make more space, and then resume processing your simulation again.

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

- Close Rocky, and then transfer all files related to your current simulation to a different drive with more space, reopen the in-process simulation, and then resume processing to this new location.

See Also:

- About Starting a Simulation (p. 908)
- About Processing Multiple Simulations In Succession (p. 920)

I get a "CPU" or "GPU" error message when I start processing my simulation

When you click the **Start** or **Resume** button to process your project, you might encounter one of the following error messages.

CPU-RELATED ERROR MESSAGES

- Fail to checkout additional CPU licenses required
- Teaching licenses can't use more than 4 cores
- Designer licenses can't use more than 8 cores

These CPU-related messages can happen when the simulation was configured to run on more CPUs than were available or allowed on the license when the simulation was started. To solve this issue, lower the **Number of Processors** value on the **Solver** | **General Settings** tab.

GPU-RELATED ERROR MESSAGES

- Teaching licenses can't use GPU
- Designer licenses can't use GPU
- No GPU enabled License available

These GPU-related messages can happen when the simulation was configured to run on more GPUs than were available or allowed on the license when the simulation was started. To solve this issue, either lower the **Target GPUs** value on the **Solver** | **General Settings** tab, or change the **Processing Unit** to CPU.

If lowering your CPU or GPU values does not solve your problem, please Submit a Support Request (p. 1293) for further assistance.

See Also:

- About Starting a Simulation (p. 908)

I got a message saying concave particles are not allowed in my Rocky version

When importing custom particles, Rocky enables you to import both convex and concave shape designs. (See also Import Custom Particle Shapes (p. 677).) However, if you want to process a simulation with concave-shaped particles, you will need a special Rocky license. If you do not already have this license when you start processing your simulation, then you may see one or both of the following messages:

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

- Simulator critical message: Concave particles are not allowed for this version
- (From the Status panel) This version of Rocky cannot use concave particles.

To resolve this issue, do one of the following:

1. Submit a Support Request (p. 1293) or contact your Rocky representative to learn about upgrading your license to include support for processing concave particle shapes.
2. When importing custom shapes, ensure you choose only convex shapes.

See Also:

- Program Limitations (p. 52)

I get an "Error simulating" or "Simulation error" message when I resume or start processing my simulation

There may be several reasons why you might get a "Error simulating" or "Simulation error" message after you resume or start processing your simulation as explained below.

THE FILE PATH OF THE SIMULATION FILE IS TOO LONG

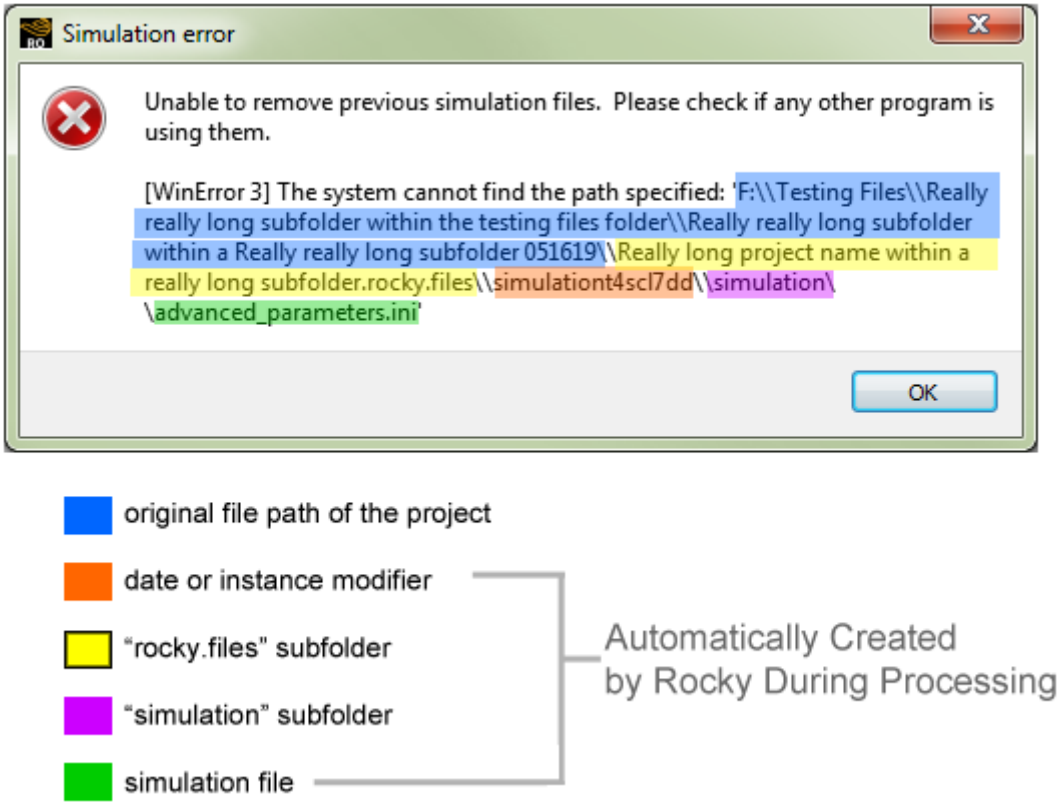
One reason you might get these messages is because the file path of the simulation file Rocky is attempting to access is too long.

The maximum number of characters that Windows allows on a full file path is 260. But due to the way Rocky creates one or more subfolders for your project to store files created during processing, it is possible for you to create a project with a shorter file path that then results in simulation files with paths that are too long for Rocky to access.

For regular Rocky projects, this includes a "rocky.files" folder that duplicates the name of your project, a "simulation" subfolder, and various output and log files (Figure 1), all of which add greatly to the character count of your original project path.

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

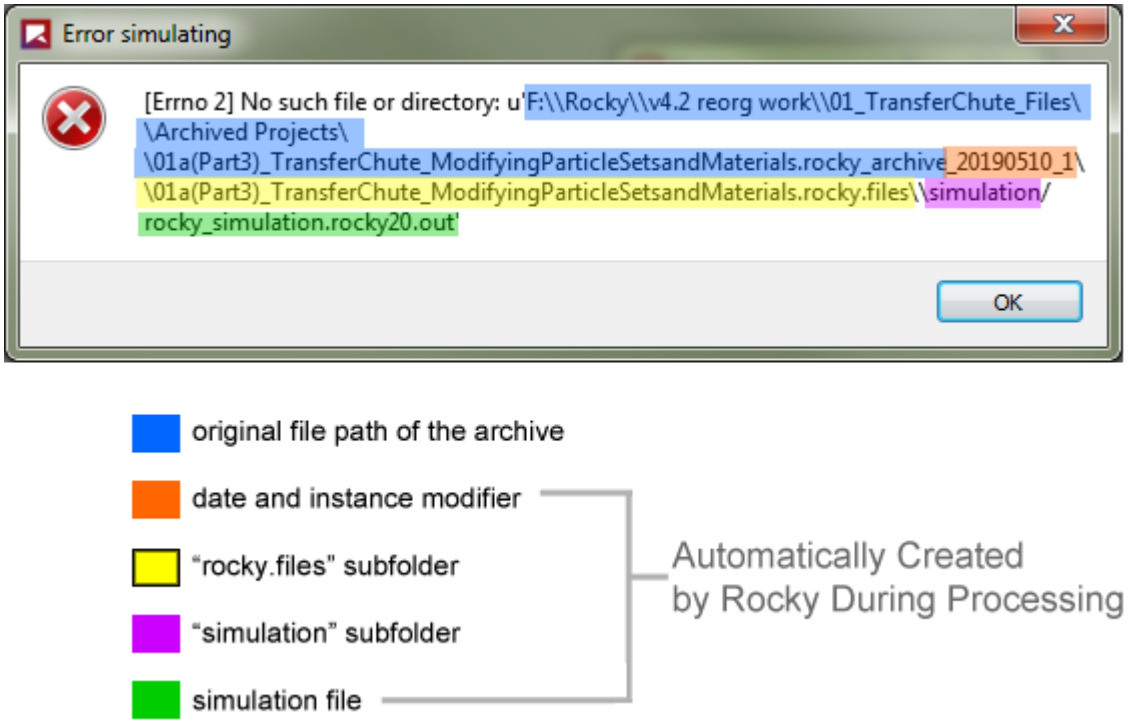
Figure 8.14: Example error when Rocky attempts to access a regular project file with a long path



This is also true for projects restored from a Rocky Archive (Figure 2).

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

Figure 8.15: Example error when Rocky attempts to access a restored archive project file with a long path



Rocky adding these additional subfolders and files after processing is why you may not be notified that your file path is too long until Rocky attempts to access the processing files within these sub-folders. That can happen during processing or when you try to resume a stopped simulation.

To resolve this issue, ensure that your initial project file path is short enough to account for the length of the subfolders and simulation files that will be created after processing.

ANOTHER WINDOWS PROGRAM IS ACCESSING THE SIMULATION DIRECTORY

Another reason you may get these messages is because another program (such as Windows Explorer) is accessing the simulation directory when you try to re-start processing a simulation from the beginning. (See also Start Processing a Simulation from the Beginning (p. 913).) This happens because Rocky erases the previous simulation directory before starting, and on Windows you cannot delete a directory if another program is using it.

To resolve this issue, ensure that other Windows programs are not accessing the simulation directory before you start processing your simulation from the beginning again.

See Also:

- I get a "Simulation setup failed" message when I start processing my simulation (p. 1345)
- About Starting a Simulation (p. 908)
- Resume Processing a Stopped Simulation (p. 913)

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

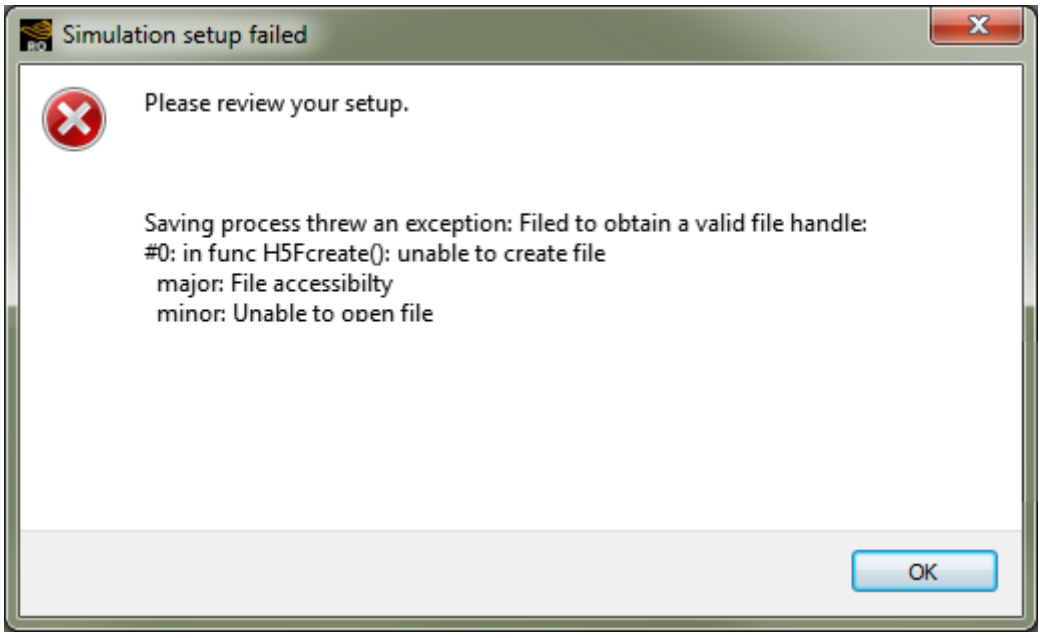
I get a "Simulation setup failed" message when I start processing my simulation

There are several reasons why you might get a "Simulation setup failed" message after you click the **Start Simulation** button to process your project, as explained below.

YOUR PROJECT FILE PATH IS TOO LONG

The maximum number of characters that Windows allows on a full file path is 260. If Rocky cannot access the simulation files that lie inside the project's directory because the path it wants is over the limit that Windows allows, it will not be able to process your simulation and will display a "Simulation setup failed" error (Figure 1).

Figure 8.16: Simulation setup failed message for long file paths



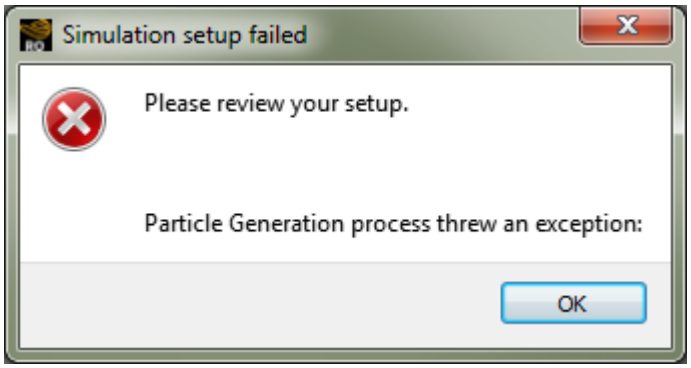
Because of this limitation, it is a good idea to stay well below the character limit when saving your project. (See also I get an "Error simulating" or "Simulation error" message when I resume or start processing my simulation (p. 1342).)

THERE ARE TOO MANY PARTICLES TO CALCULATE

If you set particle size or input parameters in such a way that results in too many particles (millions more than Rocky can handle, for example), Rocky will not process the simulation and will display a "Simulation setup failed" error (Figure 2).

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

Figure 8.17: Simulation setup failed message for too many particles



Because of this, it is best to set your particle size and particle input parameters in a way that results in no more than 20 million particles.

See Also:

- About Starting a Simulation (p. 908)

I get warnings or errors on my Simulation Log Panel

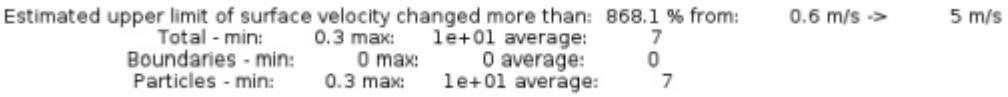
When you are processing your simulation, the Simulation Log panel displays any solver-related warnings or information that can happen during processing (see also About the Simulation Log Panel (p. 101).)

Some of the more asked-about warnings and errors are explained below. Note: This is not an exhaustive list.

SURFACE VELOCITY CHANGED

If you get a warning that says "Estimated upper limit of surface velocity changed" (Figure 1), Rocky is letting you know that a sudden change of velocity happened during the simulation. These spikes are important for you to be aware of as they could indicate that something is wrong with the forces calculations. By alerting you to these occurrences, you can better decide whether further investigation is warranted.

Figure 8.18: Example "surface velocity changed" warning on the Simulation Log panel



This message is most common during the injection phase, as particles typically enter the domain with the minimum velocity possible for the combination of inlet area, inlet time, particle mass flow, and acceleration due to gravity. In this scenario, the increase in the velocity is expected and, therefore, these messages can likely be ignored.

However, in a scenario where you do not expect huge increases in the velocity (for example, a mill operating under steady state conditions), this message can alert you to potential issues that you can then investigate. You may determine after investigation that the velocity increase is a correct simulation

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

result or you may find that it indicates numerical issues (for example, large overlaps or incorrect forces calculations) that may require changes to your simulation setup.

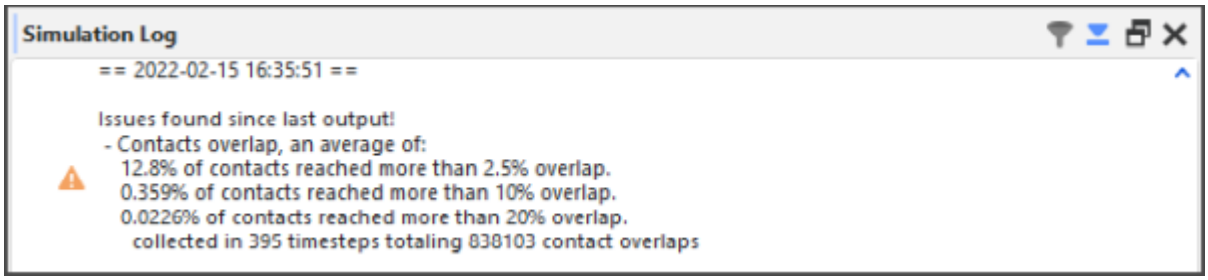
Rocky decides whether or not to show this warning-and what it displays if it does-in the following manner:

- At the beginning of the output period, Rocky has a value of the relative velocity from the previous output period. In the example case shown in Figure 1, it was 0.6 m/s.
- During the current computing output period, Rocky monitors the relative velocities. If it happens to be higher than 50% of the value it had from the previous output, it triggers the warning that velocities have increased more than 50%. In the example case above, it has computed a relative velocity of 5 m/s, which is 833% higher than the value it started with. The first line of the warning contains this info.
- If this warning was triggered, Rocky will give you the statistics of the output, at the end of this output. It will give you the minimum and maximum relative velocities for particles and boundaries for the entire output. In the example case above, the maximum relative velocity was 10 m/s and the minimum was 0.3 m/s. The maximum velocity is higher than the one printed in the first line because that one was the velocity that launched the warning, while the maximum velocity happened later in this output period.
- At the next output period, Rocky will check if the velocity is 50% higher than 5 m/s. If so, a new warning will be written to the Simulation Log.

CONTACTS OVERLAP

If you get a message about "Contacts overlap" (Figure 2), Rocky is letting you know that the number of particle-to-particle or particle-to-boundary contacts had an overlap higher than one or more of the levels defined in the **Contacts Overlap Monitor** module. The overlap percentage is determined by the size of the smallest particle in the contact pair. (See also About Monitoring Overlaps (p. 309).)

Figure 8.19: Example "contacts overlap" message on the Simulation Log panel



Monitoring your contact overlaps is important as Rocky is based on the soft-sphere approach, which uses the overlap value in order to compute collision forces. Large overlap values can lead to serious stability and accuracy issues in a simulation.

For example, when default **Overlap Levels** are used in the **Contacts Overlap Monitor** module, Rocky warns you when any overlaps higher than 2.5%, 10%, or 20% happened in the simulation, and also gives you the number of contacts that had large overlaps within that output. Typically, overlaps of 2.5% might be considered acceptable but simulations with overlaps larger than 10-20% should typically not be trusted.

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

Rocky decides whether or not to show a message or warning-and what it displays if it does-in the following manner:

- If no overlap reaches more than **Overlap Warning Level #1** (2.5% by default), no message is given.
- If any overlap reaches more than **Overlap Warning Level #1** (2.5%) but less than the other two, an info message is given.
- If any overlap reaches more than **Overlap Warning Level #2** (10% by default) but less than the other level, a warning message is given.
- If any overlap reaches higher than **Overlap Warning Level #3** (20% by default), a warning message is given.

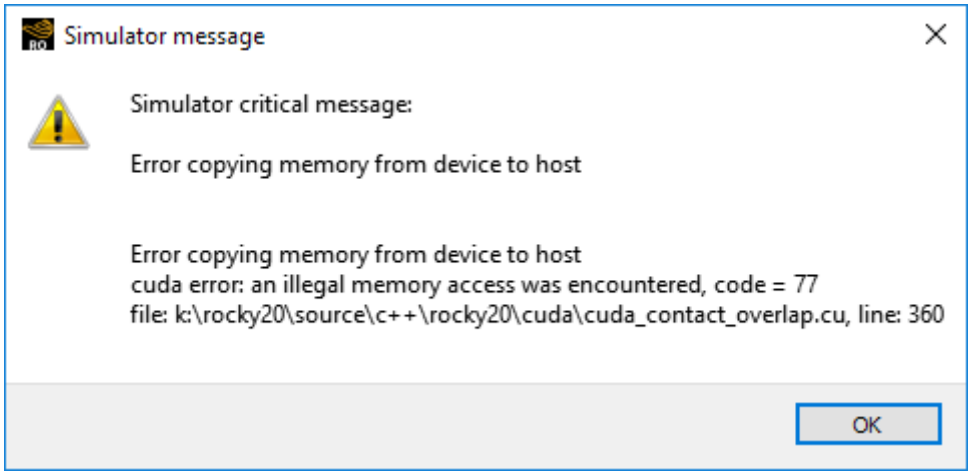
See Also:

- About the Simulation Log Panel (p. 101)
- About Monitoring Overlaps (p. 309)

I get an "Error copying memory from device to host" message when I try processing my simulation

When you click either the **Start** or **Resume** button to process your project, you might encounter a **Simulator critical message** reporting an "Error copying memory from device to host" (Figure 1).

Figure 8.20: Simulator critical message reporting "Error copying memory"



There are several reasons why you might get this error, including:

- **There isn't enough GPU memory to either start or resume the simulation.** In this case, the solution is either to adjust your case setup to require less memory, or to select a higher-memory GPU or multi-GPU resource, and then try processing again.
- **The driver for the single GPU or multi-GPU resources is outdated.** In this case, the solution is to update your GPU drivers, and then try processing your simulation again.

See Also:

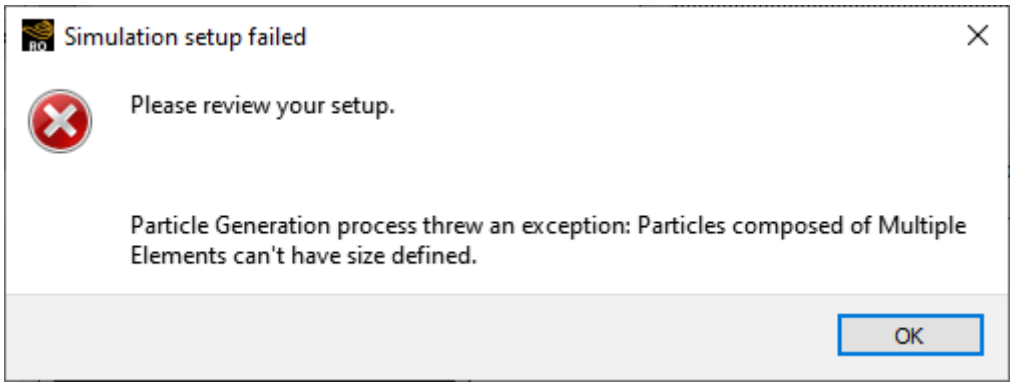
Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

- Troubleshooting Common Processing Errors (p. 1339)
- About Starting a Simulation (p. 908)

I get a "Particles composed of Multiple Elements can't have size defined" error when I try processing my simulation

When you click the **Start** button to process your project, you might encounter a **Simulation setup failed** error reporting that "Particles composed of Multiple Elements can't have size defined." (Figure 1).

Figure 8.21: Simulation setup failed error



This situation happens when you have defined a Particle set (p. 1393) with Multiple Elements (also known as meshed (p. 1392) particles) that was then used for a Custom Input whose definition CSV file includes a **Size** column. Because Rocky allows only a single size for multi-element particles, it will not support this **Size** setting for Custom Inputs.

To resolve this error, do either of the following:

- Remove the **Size** column from your Custom Input's definition file. Doing so means that only the single size you have defined for the multi-element Particle set itself will be used for all particles released by that Custom Input.
- Change the **Particle** definition for your Custom Input to instead be a Particle set made of single-element (rigid) particles.

See Also:

- Troubleshooting Common Processing Errors (p. 1339)
- About Adding and Editing Particle Inputs (p. 708)
- About Adding and Editing Particle Sets (p. 602)

I get a scripting error when I attempt to run the simulation in batch mode

If you have a Rocky simulation that you are attempting to run (either in Rocky or through Ansys Workbench) without the Rocky UI open (also known as "batch mode"), and you have scripts for tasks that require the Rocky UI to be open, you might get an error that says something similar to the following:

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

```
NotImplementedError: Error when executing scripting command: ExportAnim-
ation. Arguments: {`context_id': ` project.vizu_window_subjects_contain-
er.vizu_window_subject_00001.animation_context', `filename': `\\test.avi',
`mode': `video'}n"}
```

This might happen if you try to run scripts in batch mode for the following tasks:

- Creating and exporting an animation.
- Creating and exporting plots.
- Creating and exporting images of 3D View windows.

To resolve this issue, ensure that any scripts you are using in batch mode do not require the use of the Rocky UI.

See Also:

- Appendix G: Perform Tasks from the Command Line or Journal (p. 1308)
- About Creating and Using PrePost Scripts (p. 296)

I get a "Server returned a fault exception" error when running a script in batch mode

If you have a Rocky simulation that you are attempting to run (either in Rocky or through Ansys Workbench) without the Rocky UI open (also known as "batch mode"), and you have scripts for tasks that make use of matplotlib using Qt as a backend, you might get an error that says something similar to the following:

```
CookComputing.XmlRpc.XmlRpcFacultException: Server returned a fault ex-
ception: [1] <class `TypeError'>:_init_() missing 1 required positional
argument: `figure'
```

This can happen if the script tries to export an image when the Rocky UI is closed, which also means there is no Qt window nor context available.

To work in batch mode, you must change the script so that matplotlib uses a non-interactive backend, such as `svg'. An example of this code change is provided below:

:Command:import matplotlib

matplotlib.use(`svg')

import matplotlib.pyplot as plt`

See Also:

- Appendix G: Perform Tasks from the Command Line or Journal (p. 1308)
- About Creating and Using PrePost Scripts (p. 296)

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

8.3.2. My simulation is taking too long to process

If your simulation is taking longer than you expected to process, try doing one or more of the following:

- Run only one simulation at a time on any one machine.
- Increase your processing power and/or try running the simulation on a GPU. (See also About Starting a Simulation (p. 908).)
- Limit the particle complexity, including reducing the number of shape sides, corners, and size distributions used. (See also About Adding and Editing Particle Sets (p. 602).)
- Reduce the particle mass flow rate. (See also About Adding and Editing Particle Inputs (p. 708).)
- Make your boundary triangle sizes larger.
- Increase your Time Step Duration. (See also About the Simulation Summary (p. 859).) This can be done by increasing the size of the particles (see also Set Particle Size Ranges (p. 375)), reducing the Young's Modulus (see also About Modifying Material Compositions (p. 591)), and/or reducing the Loading N-Steps (see also About Solver Parameters (p. 464). Note: Loading N-Steps should not be set lower than 10.

See Also:

- Best Practices for Faster Processing (p. 68)

8.3.3. I set parameters for my simulation but I cannot process it

Before you are able to process a simulation, you need to set up each of the following items:

- One Particle set. (See also About Adding and Editing Particle Sets (p. 602).)
- One Input. (See also About Adding and Editing Particle Inputs (p. 708).) Note: If the input is a **Continuous Injection** type, then one inlet is also required. The inlet can either be from a default feed conveyor or from a separate inlet geometry that you have added into the simulation.

Tip:

Use the information on the Status panel to help you solve issues quickly. (See also Double-Click the Status Panel to Jump to the Appropriate UI Location (p. 117).)

See Also:

- Setting Up a Simulation (p. 411)

8.3.4. I get unusual results when I process my simulation

There are several reasons why you might be getting unusual results when you process your simulation. See below for possibilities.

- **You might have imported hidden boundaries.** When you create your boundary components in a CAD program like Ansys SpaceClaim, you have the ability to hide components or layers of com-

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

ponents. All boundaries-hidden or shown-will be imported into Rocky with an STL, XGL, or DXF 3D faces file, so it is best to remove the boundaries you do not want from the CAD program before importing the file into Rocky. This is primarily an issue when boundary walls overlap and create a case for trapped particles (see below).

- **You might have overlapping inlets.** An inlet will delay dropping particles until it detects that there are no other particles in its way. If you have set an inlet in the particle path of another inlet, this can cause the downstream inlet to produce strange spurts of particles as it starts and stops its particle flow to account for other particles in its way - **You might have trapped particles.** With moving boundaries, particles can become trapped inside a boundary wall when the new position for the boundary slices through the middle of a particle before the particle has a chance to recalculate its new position. This can cause the trapped particle to shoot out in strange directions, like popcorn. To prevent trapped particles, try the following:
  - Move boundaries slowly to give the particles enough time to re-calculate their positions.
  - Avoid overlapping boundary walls.
  - As a last resort, select the **Release Particles without Overlap Check** checkbox on the **Solver | General** tab. (See also About Solver Parameters.)
- **Your simulation domain might be too small for your particle shapes.** This happens most frequently with fiber (p. 1391) shaped particles. (See also A fiber or other particle shape disappeared from my simulation during processing (p. 1357).)

If after taking the steps above you still get unusual results, please contact support for assistance. (See also Getting Help and Support (p. 1285).)

See Also:

- Troubleshooting (p. 1315)

8.3.5. I am processing my simulation but no output files are being created

If you have clicked **Start** on the **Solver** tab but see that no output files (p. 1393) are being created, then one of the following events might be happening:

- You have not yet updated the results using the **Refresh** button or the **Auto Refresh** checkbox on the progress toolbar. (See also Start Processing a Simulation from the Beginning (p. 913).)
- Your simulation is taking longer to process due to its complexity and/or a limited amount of processing power. (See also My simulation is taking too long to process (p. 1351).)
- The **Simulation** Output Frequency might not be set low enough for the **Simulation Duration** you specified. In order for output files to be created, the Simulation Output Frequency value must be smaller than the Simulation Duration value. (For example, a Simulation Output Frequency of 5 for a Simulation Duration of 20 will yield 4 output files.) To lower the Simulation Output Frequency, do the following:
  1. From the **Data** panel, select **Solver**.
  2. From the **Data Editors** panel, select the **Solver | Time** tab.

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

3. In the **Simulation** Output Frequency box, enter a smaller number. (0.05 is the recommended default.)

4. Try processing your simulation again. (See also Start Processing a Simulation from the Beginning (p. 913).)

See Also:

- About Solver Parameters (p. 464)

8.3.6. My particles are flowing too quickly

The speed by which the particles flow out of an inlet with continuous injection input is based upon the following criteria:

- The particle mass flow rate of the input.
- The size of the inlet.
- The injection period of the input.

In general, the smaller your inlet and the shorter your injection period, the faster the particles will flow out of it. To have your particles flow slower, make your inlet larger, increase your injection period, and/or reduce your particle mass flow rate.

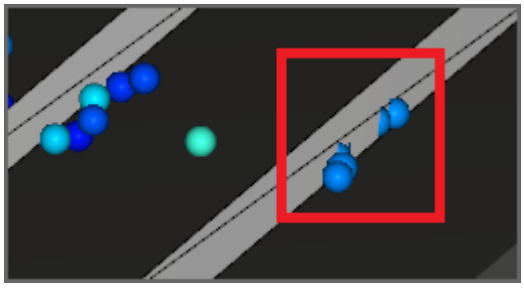
See Also:

- About Adding and Editing Particle Inputs (p. 708)
- About Inlet Parameters

8.3.7. Particles are going through the walls of my geometry

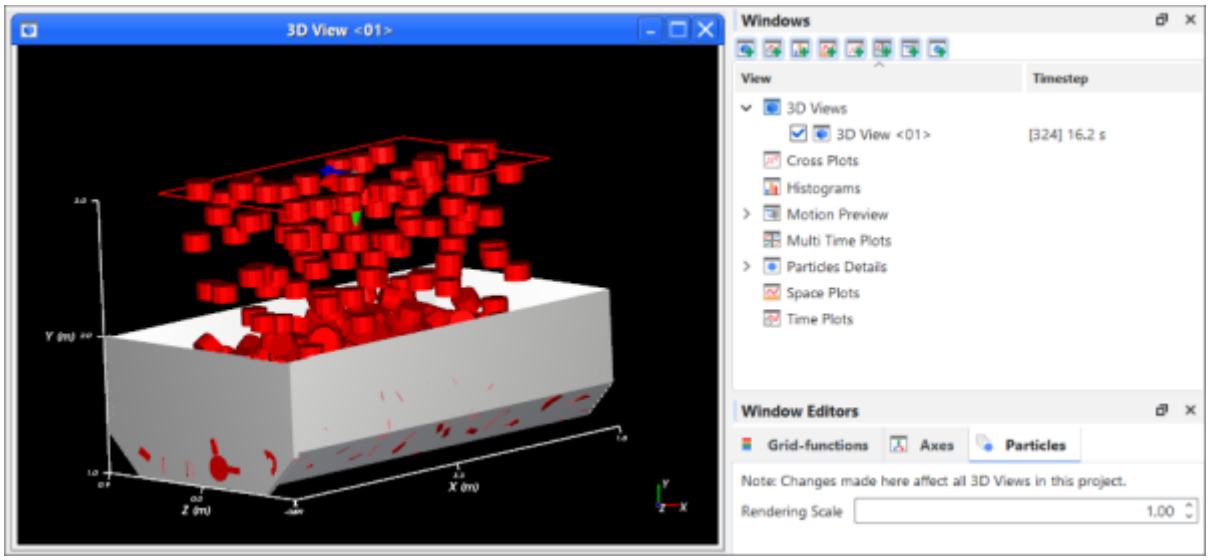
In Rocky, there are several events that may cause your particles to appear as if they are going through the walls of a geometry during a simulation, examples of which are shown in Figures 1 and 2.

Figure 8.22: Example showing simulated Sphere particles going through a geometry wall



Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

Figure 8.23: Example showing simulated particles "bleeding" through a geometry wall



The most common reasons and possible solutions are discussed below.

Periodic Cartesian Domain Issues

In this version of Rocky, it is possible for you to set the limits of your Periodic Cartesian Domain to be smaller than your geometry limits and/or the coordinate limits of your domain. In doing this, however, you run the risk of having particles go through the walls of your geometries during the simulation.

This is likely to happen if a geometry has triangles with all three vertices lying exactly on the plane that defines the periodic limits in a direction; those triangles will not contact the particles even if the boundary motion leads the triangles away from the periodic limits. This can cause the particles to go right through the geometry.

To solve this, change the limits of your periodic domain to avoid having them line up exactly with the triangles of your geometry walls. (See also About Domain Settings Parameters (p. 452).)

Particle Velocity Issues

If the velocity of a particle is too high given both the time step and the stiffness of the item it contacts, it is possible for a particle to shoot right through the geometry. This can happen for any case where the relative velocity of two particles colliding or a particle and and a boundary colliding are high. For example:

- When the initial particle injection speed is too high.
- When the particle injection speed is low but you have a boundary accelerating or perhaps rotating the domain very fast, which during contacts can fling the particles through boundary walls.

Particle injection speed is based upon the particle mass flow rate, and the size of the inlet, and injection period of the input. So in general, to reduce your injection speed, make your inlet larger, increase your input injection time, and/or reduce your particle mass flow rate. (See also My particles are flowing too quickly (p. 1353).)

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

To slow down fast-moving geometries, you can adjust your velocity parameters using Motion Frames. (See also About Creating and Applying Motion Frames (p. 533).)

Rolling Resistance Issues

When used with shaped particles (p. 1395), rolling resistance can cause strange behavior, including having particles go through boundaries and other unwanted behavior. It is therefore best to set Rolling Resistance to zero in these cases:

- When you are using elongated particles (i.e., shapes with high aspect ratios).
- When you have set your Tangential Force model to be Mindlin-Deresiewicz. (This is due to a known issue that could lead to an unstable Angle of Repose. See also Physics and Force Limitations (p. 55).)

(See also About Adding and Editing Particle Sets (p. 602).)

Geometry Thickness Issues

A visual overlap between the particles and the geometry can occur, for example, when the geometries are imported as a surface. Because a surface boundary has no thickness, the particles can sometimes appear to come through (or "bleed" through) it (Figure 2). This affects only the visualization of the case and has no effect upon the results nor analyses.

To resolve this issue, try lowering your Particle Rendering Scale. (See also About Using the Window Editors Panel to Change All 3D Views in the Project (p. 1048).)

See Also:

- About Domain Settings Parameters (p. 452)
- My particle size is too large for my periodic domain (p. 1324)

8.3.8. No particles appear when I process the simulation

If your simulation appears to be processing and is generating *output files* (p. 1393) but you are seeing no particles appearing on your screen, one of the following events might be happening:

- You haven't yet created a 3D View window, which is the method by which you visualize geometries and particles. (See also View Geometries, Particles, Points, and Fluids in 3D (p. 1041).)
- You have disabled the **Particles** entity (p. 1395) on the **Data** panel. (See also Show/Hide Components by Using Eye icons and Checkboxes (p. 113).)
- The particles have been set as a **Solid color** the same color as the 3D View background, rendering them invisible. (See also About Using the Coloring Tab to Change a 3D View (p. 1074).)
- No particles have entered the domain yet. This might happen if the continuous injection input has not yet become active, perhaps due to the start and/or stop time settings, or because the mass flow rate is set low enough that there is a long break between individual particles being released (see also About Adding and Editing Particle Inputs (p. 708)). In this case, it is possible that you happen to be viewing a time step where particles have not been released yet, and waiting for further time steps to be calculated might resolve the issue.

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

- You have not yet updated the results using the **Refresh** button or the **Auto Refresh** checkbox on the progress toolbar. (See also Start Processing a Simulation from the Beginning (p. 913).)

See Also:

- About Adding and Editing Particle Sets (p. 602)

8.3.9. Particles stopped flowing before my simulation was done processing

If at any point during the simulation, a particle cannot completely enter the simulation area, no more particles will enter the simulation and the particle flow will stop. A particle can be prevented from entering the simulation area by a boundary or another particle.

For example, if a chute gets blocked and particles back up close enough to the entry point (an inlet or the top of a feeder box, for example), the particle flow will stop. Or, for example, if the top of a feeder box is sloped enough to not allow enough space for a particle to fully enter at the corners, the particle flow will stop.

To fix this issue, try one or more of the following solutions:

- Add a vertical lip to the top of your feeder box.
- Make the inlet smaller or higher.
- As a last resort, select the **Release Particles without Overlap Check** checkbox on the **Solver** | **General Settings** tab. (See also About Solver Parameters (p. 464).) Note: Turning off the overlap check can lead to unusual results as overlapped particles might add extra energy to the particle flow, which can affect other particles and boundaries.

See Also:

- Best Practices for Setting up Geometries (p. 70)

8.3.10. I cannot find a batch mode for processing

Rocky comes with a separate program called Rocky Scheduler that is used to process multiple simulation projects at once. The Rocky Scheduler program is saved to the same directory as the main Rocky program. (See also Open the Rocky Scheduler Program (p. 927).)

Batch mode may also refer to running Rocky through the command line. (See also Appendix G: Perform Tasks from the Command Line or Journal (p. 1308).)

See Also:

- About Processing Multiple Simulations In Succession (p. 920)

8.3.11. I want to use a Pascal-based GPU card instead of the recommended CUDA-based one

Pascal-based GPUs are still CUDA-based, so they will work just fine for processing in Rocky. The Pascal title refers to the architecture of the GPU, as Maxwell did to the previous generation. Maxwell was 28 nm and Pascal is 16 nm chip size.

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

See Also:

- About Starting a Simulation (p. 908)

8.3.12. I cannot find where to set the number of cores for CPU processing

You set the number of cores you want to use during processing in the **Number of Processors** box, which is located on the **Solver** | **General** tab. (From the **Data** panel, click **Solver** and then from the **Data Editors** panel, select the **Solver** tab and then the **General** sub-tab.)

See Also:

- About Starting a Simulation (p. 908)

8.3.13. My GPU card is not being recognized as CUDA compatible

If you see a notification from Rocky that your GPU card is not CUDA compatible, and you know your card is supported by Rocky (see System Requirements (p. 67) for a list of supported GPU cards), then you have most likely installed a new version of Rocky that requires a newer driver for your GPU card. Even if you have automatic updates set and your device manager shows your card as up-to-date, you may still benefit from a manual driver update. (See Best Practices for Installing a New Version of Rocky (p. 68) for specific instructions.)

See Also:

- About Starting a Simulation (p. 908)
- Set Up a New GPU Card (p. 404)

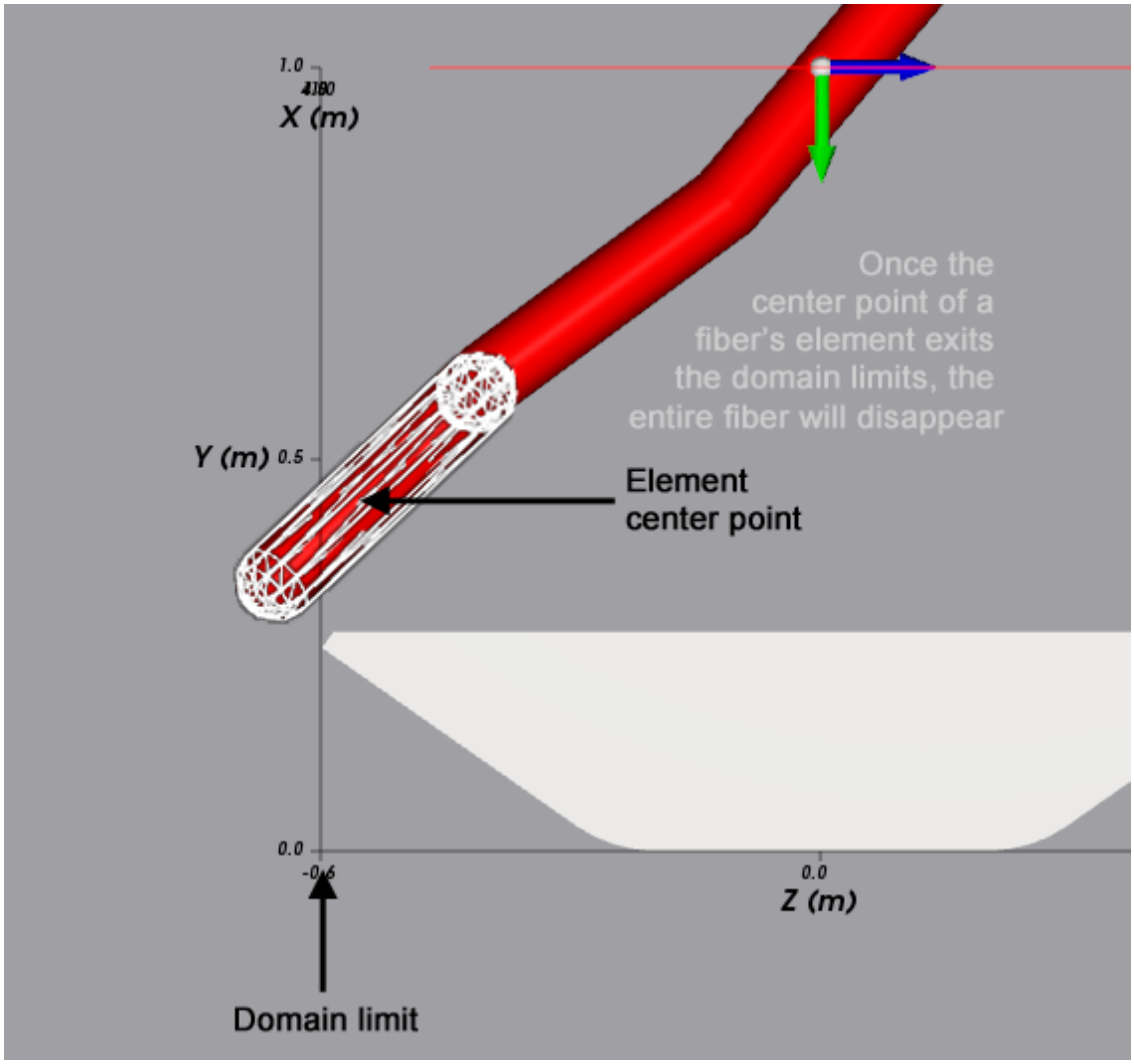
8.3.14. A fiber or other particle shape disappeared from my simulation during processing

If during your simulation, you notice that a fiber (p. 1391) or other particle shape disappears suddenly from your screen, it is most likely caused by the particle reaching the limits of the domain.

In Rocky, all particle shapes exit the domain as soon as the center of the shape reaches the domain limits. Because fibers are made up of many separate smaller shapes called elements (p. 1390), it is the center of the element and not the fiber shape itself that is counted in these cases. This means that even if only a small portion of the fiber reaches the domain limits, the whole fiber will disappear (Figure 1).

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

Figure 8.24: A fiber particle just before the center of one of its elements reaches the domain limit and disappears from the simulation



Because of this key difference, disappearing fiber particles are much more noticeable, and could happen with more frequency than with other particle shapes.

It is therefore important that when simulating fiber shapes you pay extra attention to the size of your domain. Ensuring that your domain is large enough to support the extra length of a fiber particle will help ensure that your particles do not disappear.

See Also:

- About Domain Settings Parameters (p. 452)
- About Adding and Editing Particle Sets (p. 602)
- Troubleshooting Simulation Processing Problems (p. 1338)

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

8.3.15. I cannot resume processing my older simulation in the latest version of Rocky

You may only resume processing your stopped simulation if the simulation was last processed in the same major version of Rocky as the version you are using now. For example, if you started processing your simulation in version 4.1.0, you may resume processing it in any 4.1.x version released thereafter, but may not resume it in any version of the next major release of 4.2.x, and so on.

For these latter situations, you must start processing the project again from the beginning. (See also Start Processing a Simulation from the Beginning (p. 913).)

See Also:

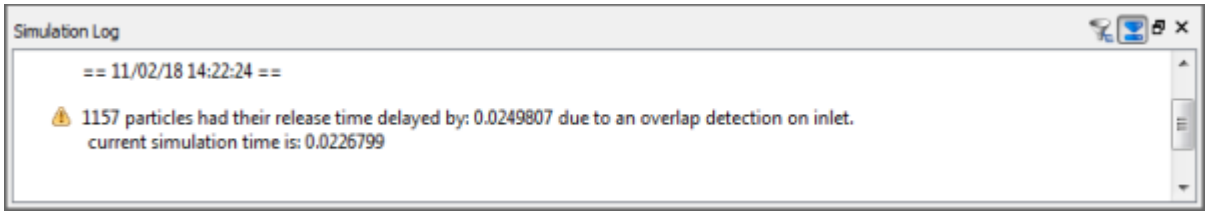
- Resume Processing a Stopped Simulation (p. 913)

8.3.16. My inlet stopped injecting particles for a long time and then later resumed it

While processing a simulation, Rocky's continuous injection algorithm for inputs tries to release particles into the simulation without causing any particles to overlap other particles. If Rocky predicts that an overlap will happen, it will hold off injecting the next layer of particles for a period of time, then it will try again. If an overlap is still predicated at that later point, it will hold of releasing the layer again for another period of time, and so on.

When this kind of injection delay happens, Rocky will provide you with a warning in the **Simulation Log** panel (Figure 1) as "N particles had their release time delayed by X due to an overlap detection on inlet".

Figure 8.25: Example overlap warning found on the Simulation Log panel



If you find Rocky is waiting too long to release particles, you can try adjusting the following parameters:

- The injection waiting period. In Rocky, this is called **Overlap Particles Delay** and can be found on the **Solver | General** tab. The default value is 0.025 s. (See also About Solver Parameters (p. 464).)
- The size and location of your inlet. This can help provide more space and simulation time for particles to be injected.
- The mass flow rate and injection timeframe of your continuous injection input. This can help reduce the amount of particles that are released. (About Adding and Editing Particle Inputs (p. 708).)
- The shape, size, and/or orientation of your particle set (p. 1393). (See also About Adding and Editing Particle Sets (p. 602).)

See Also:

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

- About the Simulation Log Panel (p. 101)
- I get unusual results when I process my simulation (p. 1351)
- Particles stopped flowing before my simulation was done processing (p. 1356)

8.3.17. My Volumetric Inlet isn't working right

While the Volumetric Inlet method for Particle Input has several options that provides maximum flexibility with regards to your simulation set up, this complexity can also increase your chances for mistakes.

Use this topic to learn about the most common issues and how to avoid them.

Fewer particles were injected than estimated on the Info tab

For Volumetric Inlet, Rocky provides a particle estimate on the **Info** tab assuming that there are no boundary limits to consider, and assuming a **Gap Scale Factor** equal to **1**. This means that you can set a **Mass** much higher than the box and gap you define will hold, but Rocky will not know that until after the simulation is processed.

To avoid this, ensure that you are entering a Mass value appropriate for the Particle set, gap factor, and box volume.

Only a few (or no) particles were injected

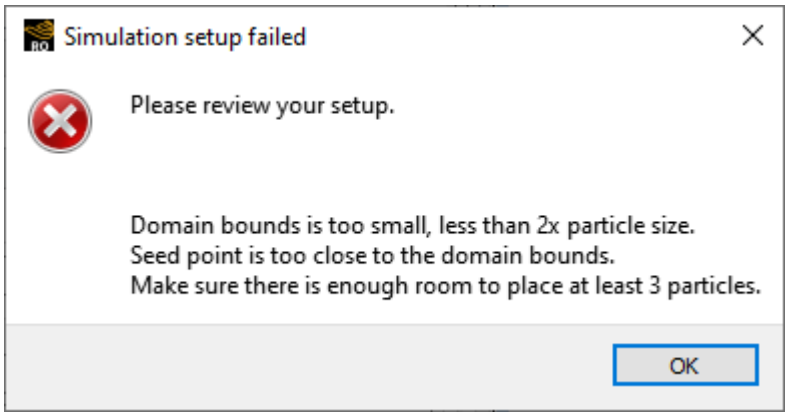
If the **Seed Coordinate** of your Volumetric Inlet happens to land directly on the bounds of a geometry component, Rocky may have trouble calculating the fill due to overlaps with the boundary.

To avoid this, ensure that you locate your **Seed Coordinate** well away from the boundaries of your simulation geometries.

No particles were injected

If your Volumetric Inlet properties result in fewer than 3 particles, the Volumetric Inlet will not work, you will get no particles injected into your simulation, and you might get an error when you try to process your simulation (Figure 1).

Figure 8.26: Simulation setup failed error



Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

Volumetric Inlet Inputs require at least 3 particles in order to create the "Seed".

To fix this, increase your **Mass**, reduce your **Particle Size**, reduce your **Gap Scale Factor**, and/or make your fill area larger to accommodate more particles.

Tip:

To see an estimate of how many particles Rocky expects to generate given the parameters defined, view the information on the **Info** tab for the Volumetric Inlet Input you are defining. Note: Keep in mind that the estimate assumes there are no boundary limits to consider and assumes a **Gap Scale Factor** equal to **1**. This means that if you have set your particle **Mass** higher than the box and gap you define can contain, the particle estimate on the **Info** tab will show a higher number than what will actually be injected.

See Also:

- About Adding and Editing Particle Inputs (p. 708)
- Create a New Volumetric Inlet Input for Particles (p. 741)
- About the Info Tab (p. 83)

8.3.18. I cannot change my setup parameters during processing

Most setup parameters (see also see also Setting Up a Simulation (p. 411)) you are unable to change while your simulation is actively processing. This is because changing those parameters would have an effect upon the calculations that are already in process.

You will know that a parameter is not able to be changed during processing when it appears disabled (i.e, grayed out) in the UI.

The one exception to this rule is for the setup options on the top most Study entity (see also About Study Parameters (p. 415)), which can be changed at any point, even during active processing. This is because while these settings do affect the project file, they do not affect the underlying calculations being performed.

To change any of the other setup parameters, you must first stop your simulation (see also Stop a Simulation from Processing (p. 915)). At that point, the only other parameters besides the Study entity ones that you can modify without invalidating your results are the **Execution** options on the **Solver** | **General** tab, and the options on the **Solver** | **Advanced** tab (see also About Solver Parameters (p. 464)). If you change any of these parameters while your simulation is stopped, you will still be able to Resume processing (see also Resume Processing a Stopped Simulation (p. 913)) after making your changes.

Any other setup parameters that you want to modify will require you to either delete your results or make a copy of your project setup before you will be able to change them (see also Setting Up a Simulation (p. 411)). It is therefore important to ensure that your settings are correct before you begin processing.

While it is not possible to change most of your setup parameters during active processing, you should still be able to create and modify non-setup parameters such as those for data viewing and analysis (see also Analyzing a Simulation (p. 931)). This includes User Processes, Properties, Curves, and windows such as 3D Views and plots. These all make use of the data Rocky calculates during processing-and

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

will therefore be limited to only the data that has been processed-but themselves have no affect upon how the calculations are made. Therefore, you are able to make use of these forms of analysis during active processing.

See Also:

- Processing a Simulation (p. 907)

8.3.19. My particle's height increases after each collision

If you have a particle that when simulated, drops from a certain height, bounces, and then later bounces higher-as if the energy is increasing over time-you have likely defined the following parameters in your simulation:

- A non-round particle shape with **Enable Rotations** turned on (default).
- The interaction between the material used for the particle and the material used for the boundary it is colliding with has a **Restitution Coefficient** between 0.5 and 1.
- The time step (p. 1395) is calculated automatically (default).

In this particular situation, the particle height might seem to increase after each collision. This is because the time step that Rocky automatically uses by default is not small enough to accurately simulate rotating non-round particles that have no (or almost no) dissipation. (No dissipation is represented by a **Restitution Coefficient** equal to 1.)

In Discrete Element Modeling (DEM), the dissipation is rarely zero and, therefore, the restitution should rarely be equal to 1. Rather, typical restitution should be less than 0.5 and the default time step works fine for these cases.

For restitutions above 0.5, the default time step is no longer sufficient and if left as default, will cause errors to accumulate in the simulation. The closer to 1 is your restitution value, the more errors are expected to accumulate, which can result in the inaccurate particle behavior described above.

The best way to prevent errors from accumulating in these situations is to decrease the time step. The closer the restitution value is to 1, the more significant the time step decrease should be.

Another thing to take note of in this situation is the size of the triangular mesh of the colliding boundary in relation to the particle size. If the triangular mesh is significantly smaller than the particle, it creates multiple contacts for the impacting particle, decreasing the accuracy of the simulation. If you increase the size of your triangles so that the particle impacts just one triangle at a time, for example, you will get more accurate results.

In summary, to resolve this issue, try one or more of the following:

- Use **Spheres** or other rounded particle shapes. (See also About Adding and Editing Particle Sets (p. 602).)
- Turn off **Enable Rotations**. (See also About Adding and Editing Particle Sets (p. 602).)
- Set the **Restitution Coefficient** values to between 0.1 and 0.5. (See also About Modifying Materials Interactions and Adhesion Values (p. 597).)

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

- Decrease the time step. (See also About Solver Parameters (p. 464).)
- Increase the size of your colliding boundary's triangular mesh.

See Also:

- About Adding and Editing Particle Sets (p. 602)
- About Modifying Materials Interactions and Adhesion Values (p. 597)
- About Solver Parameters (p. 464)

8.4. Troubleshooting Simulation Analysis Problems

What kind of problem are you having?

- I cannot find the zoom, pan, or tilt options on the toolbar (p. 1364)
- I cannot see the data or statistics (p. 1364)
- I cannot view my previous output files (p. 1364)
- I cannot remove the colored box around my 1-Way LBM or User Process selection (p. 1364)
- I cannot get a time counter to appear on my simulation (p. 1365)
- I cannot get my company logo to appear on my simulation (p. 1366)
- I cannot find the minimize, restore down, and close buttons for my maximized Rocky window (p. 1366)
- I do not understand how power is calculated (p. 1367)
- I want to analyze Property information for only one of my Particle sets (p. 1368)
- I do not understand how my Particle set names map to the Particle Group value numbers (p. 1368)
- I cannot find the Particle Name nor Particle Input Property (p. 1369)
- I cannot find some Geometry Properties or Curves in this version of Rocky (p. 1369)
- Parts of my geometries and particles disappear when I rotate my 3D View (p. 1374)
- I cannot display or plot a particular property (p. 1375)
- My old script no longer works (p. 1375)
- I still want to use my old macros (p. 1376)
- My trajectory lines are outside my periodic domain (p. 1376)
- I have large discrepancies in my Intra-particle Collisions Statistics Frequency values (p. 1378)
- I cannot find the directional properties for Solid Implicit Force (p. 1379)

See Also:

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

• Troubleshooting (p. 1315)

8.4.1. I cannot find the zoom, pan, or tilt options on the toolbar

In this version of Rocky, the only way to zoom, pan, or tilt the image in the 3D View window is to use the mouse.

See Also:

• About Using the Mouse and Keyboard to Change the View (p. 1043)

8.4.2. I cannot see the data or statistics

In Rocky, some data is collected automatically and some data you must know to first opt in to collecting before you process your simulation. In addition, some data is always collected right from the start of the simulation, and some data waits to be collected until passing certain thresholds that you define.

To help you better understand how and when data is calculated, refer to the About Collecting Data in Rocky (p. 306) topic.

8.4.3. I cannot view my previous output files

If you have saved a copy of your partially processed simulation (see also Save a Copy of a Partially Processed Simulation for Restart Purposes (p. 855)) and have opened it for the purpose of adding or modifying setup parameters (see also About Changing Geometries and/or Particles in a Partially Processed Simulation (p. 847)), the copy's first [zero (0)] output file will have the same data as the output file you had selected on the original when you saved the copy, but not the time.

For example, if on your original simulation you save a copy 5 seconds into the simulation and particles were just coming onto the receiving belt, the copy will look just like that 5 second file but the time will be reset to zero (0). This means that you will be unable to review prior to that point within the copy. If you need to see what was happening prior to the point at which you saved the copy, you will have to go to the original simulation to view the original output files.

See Also:

• About the Time Toolbar (p. 1279)

8.4.4. I cannot remove the colored box around my 1-Way LBM or User Process selection

When you select 1-Way LBM or a User Process from the **Data** panel, the 3D View will show a colored box around the selection, as follows:

• A white box indicates that an entity is both visible and is currently selected. To remove the white box from being shown in the 3D View, just select another entity from the **Data** panel.

• A blue box represents the boundaries of the User Process domain and will persist as long as the entity is selected - even if the entity is hidden. Like the white box, the blue box will also be removed from being shown in the 3D View when you select another entity from the **Data** panel.

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

A green box represents the boundaries of the 1-Way LBM domain. As long as the entity it represents is visible (either through the **Visible** checkbox on the **Coloring** tab or through the **Data** panel Eye icons), and not currently selected, the green box will be shown in the 3D View. The only way to remove the green box from the 3D View is to disable or make hidden the entity it represents.

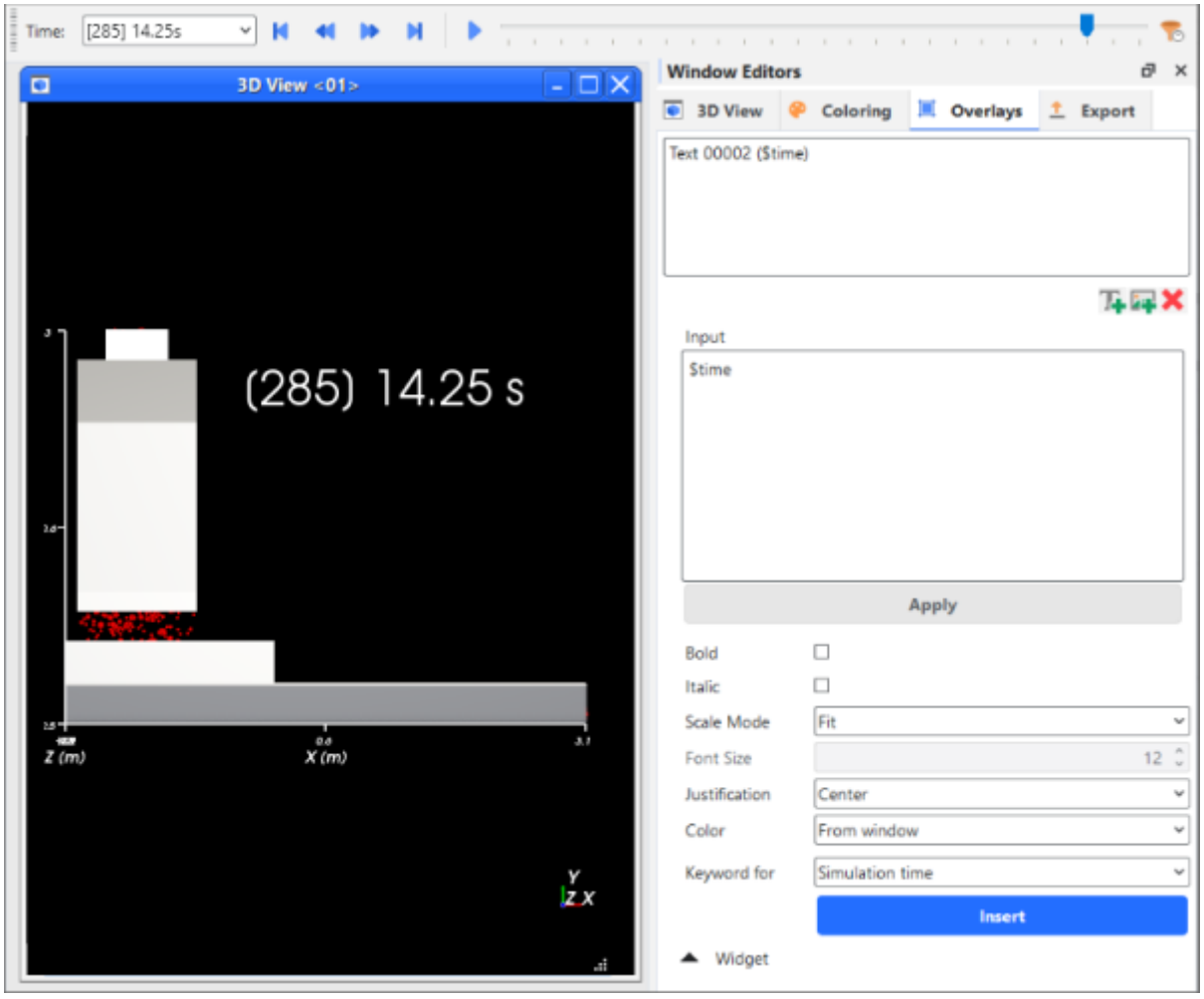
See Also:

- Create and Modify a 3D View (p. 1040)
- Show/Hide Components by Using Eye icons and Checkboxes (p. 113)

8.4.5. I cannot get a time counter to appear on my simulation

By default, no text overlays are shown on your 3D View windows. If you want to display the current time step on a 3D View window, you must add a new text overlay, and then insert **Simulation time** as the keyword you want to display (Figure 1).

Figure 8.27: Simulation time keyword added to a 3D View as a text overlay



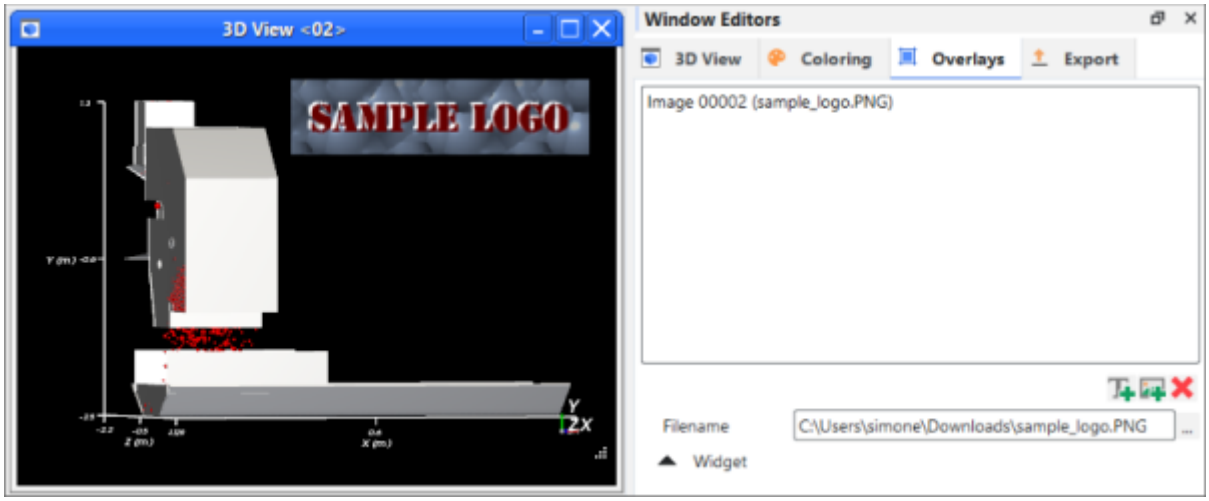
See Also:

- Add or Edit a Text Overlay for a 3D View Window (p. 1069)

8.4.6. I cannot get my company logo to appear on my simulation

By default, only the Rocky logo appears on your 3D View windows. If you want to add an additional logo, you may do so by adding an image overlay, and then changing the size and position of the overlay as you wish.

Figure 8.28: Sample logo added to a 3D View as an image overlay



See Also:

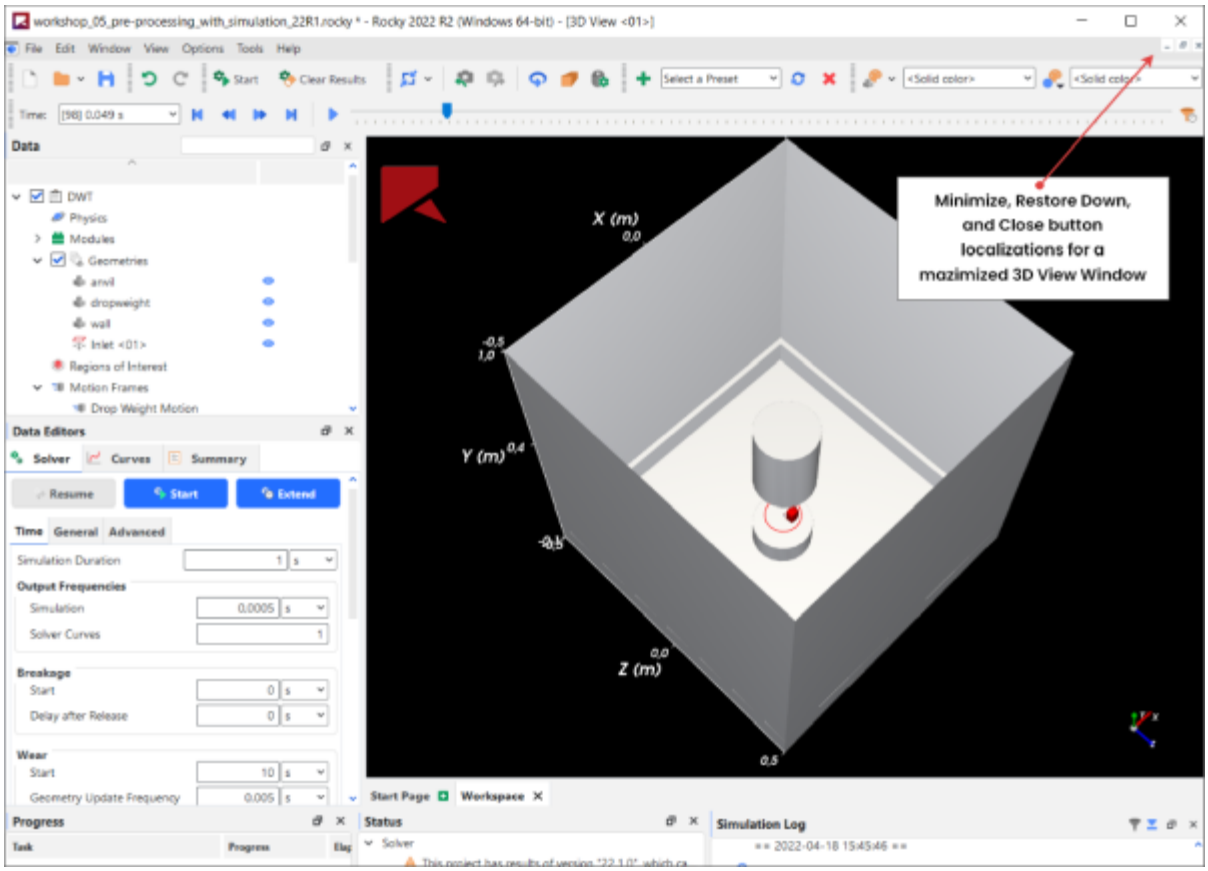
- Add or Edit a Text Overlay for a 3D View Window (p. 1069)

8.4.7. I cannot find the minimize, restore down, and close buttons for my maximized Rocky window

When you maximize a 3D View, Plot, or Histogram window, the Minimize, Restore Down, and Close buttons are located directly beneath the main program buttons (Figure 1).

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

Figure 8.29: Button locations for a maximized window in Rocky



8.4.8. I do not understand how power is calculated

In this version of Rocky, there are many ways that power information can be collected, calculated, and analyzed-in both Properties and Curves.

Please refer to the Rocky Technical DEM Manual for details on the different ways power can be calculated. (From the **Help** menu, point to **Manuals**, and then click **DEM Technical Manual**.)

Tip:

From within the PDF, you can use your viewer's Search or Find functionality to find the topics you are looking for.

See Also:

- About Collecting Data in Rocky (p. 306)
- About Properties (p. 933)
- About Curves (p. 989)
- Enable and View Power Draw Data

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

8.4.9. I want to analyze Property information for only one of my Particle sets

By default, when you select **Particles** from the **Data** panel, any information you view on the **Properties** tab applies to all Particle sets you have created. If you are interested in seeing Property information for only one of your Particle sets, you must first create a Property User Process, and then use that process to filter by **Particle Group**. By viewing the Properties for the resulting User Process, you are able to see only the information for the particle group you selected.

Note:

A particle group is the same as a Particle set (p. 1393).

To create a Property User Process that limits by Particle Group:

1. From the **Data** panel, right-click **Particles**, point to **Processes**, and then choose **Property**. A new Property entry appears under User Processes in the Data panel.
2. From the **Data Editors** panel, select the **Property** tab.
3. From the **Property** list, select **Particle Group**.
4. Do one of the following:
   - To select a single particle group, from the **Type** list, select **Value** and then in the **Cut value** box, enter the number corresponding to the particle group you want.
   - To select a range of particle groups, from the **Type** list, select **Range** and then enter what you want in the **Minimum value** and **Maximum value** boxes.

Tips:

- Particle group values are given in the order by which you have added particle groups, starting with zero. To determine what number to enter, show **Particle Groups** for the Particles entity in a 3D View window, and use the information on the legend provided. (See also I do not understand how my Particle set names map to the Particle Group value numbers (p. 1368).)
- To revert values back to the default settings, which uses the Property limits to calculate middle-range values, click the **Reset** button.

See Also:

- About the Property User Process (p. 1202)

8.4.10. I do not understand how my Particle set names map to the Particle Group value numbers

A single particle set (p. 1393) includes the shape type, size distribution, and breakage parameters. In this version of Rocky, a particle set is equal to a Particle Group.

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

The easiest way to determine how the numbers assigned to the particle groups map to the Particle sets you have defined is to view the Particle Groups Property in a 3D View window, and then use the information provided on the legend to determine the mapping.

To view the legend for the Particle Groups Property:

1. From the **Data** panel, select **Particles**.
2. From the **Data Editors** panel, select the **Properties** tab.
3. Under **Name**, locate **Particle Group** and then click-and-drag it to an empty location on the workspace. A new 3D View window will appear with the Particle set data and legend information in the **V - Name** format where **V** is the particle set value Rocky assigned and **Name** is the name of the Particle set used. Note: If you have set multiple release points for a single Particle set, then you will see multiple values of the same **Name**. For this reason, filtering by Particle Group may not be as useful for simulations involving multiple inlets that are releasing the same Particle set.

See Also:

- I want to analyze Property information for only one of my Particle sets (p. 1368)

8.4.11. I cannot find the Particle Name nor Particle Input Property

Both the **Particle Name** and **Particle Input** properties for Particles (see also About Properties (p. 933)) were removed as of the v4.2 release of Rocky.

You should be able to get the same information by using the **Particle Boundary** properties instead.

See Also:

- I do not understand how my Particle set names map to the Particle Group value numbers (p. 1368)

8.4.12. I cannot find some Properties or Curves in this version of Rocky

As of Rocky v4.5.0, one Solver Curve was removed from the product that was called "CFD Computation Fraction". (See also About Curves (p. 989).)

To review changes from prior versions of Rocky, review the sections below.

Changes Prior to v4.5.0

As of Rocky v4.4.0, some existing Properties and Curves for Geometries were combined into new Properties and Curves, some were renamed, and some were removed. Use this topic to find the data that you are seeking.

CHANGES AFFECTING GEOMETRIES

(See also Add and Edit Geometry Components (p. 480).)

Combined Geometry Properties The following two Properties were combined into a single new property called "Intensity : Dissipation":

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

- Intensity : Dissipation : Normal
- Intensity : Dissipation : Tangential Note: In versions prior to v4.3.0, this property was known as "Instantaneous Shear Power".

Combined Geometry Curves The following two Curves were combined into a single new Curve called "Power : Dissipation":

- Power : Dissipation : Normal
- Power : Dissipation : Tangential Note: In versions prior to v4.3.0, this curve was known as "Shear Intensity".

CHANGES AFFECTING PARTICLES

(See also About Adding and Editing Particle Sets (p. 602).)

Combined Particle Set Properties (Per Set Only)

- The following two Properties were combined into a single new Property called "Intensity : Dissipation":
  - Intensity : Dissipation : Normal Note: In versions prior to v4.3.0, this property was known as "Intensity Normal".
  - Intensity : Dissipation : Tangential Note: In versions prior to v4.3.0, this property was known as "Intensity Tangential".
- The following two Properties were combined into a single new Property called "Power : Dissipation":
  - Power : Dissipation : Normal (now known only as "Power : Dissipation")
  - Power : Dissipation : Tangential (now known only as "Power : Dissipation")

Combined Particle Curves The following two Curves were combined into a single new Curve called "Energy : Dissipation":

- Energy : Dissipation : Normal
- Energy : Dissipation : Tangential

Renamed Particle Curves The Curves that were renamed include the following:

- Specific Power (Normal) (now known as "Cumulative Specific Power : Impact")
- Specific Power (Shear) (now known as "Cumulative Specific Power : Shear")
- Power (Normal) (now known as "Power : Impact")
- Power (Tangential) (now known as "Power : Shear")
- Collision Rate (Normal) (now known as "Rate : Impact")
- Collision Rate (Tangential) (now known as "Rate : Shear")

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

Changes Prior to v4.4.0

As of Rocky v4.3.0, some existing Properties and Curves for Geometries, Motion Frames, Particles, and Intra-Particle Collision Statistics were renamed and some were removed. Use this topic to find the data that you are seeking.

CHANGES AFFECTING GEOMETRIES

(See also Add and Edit Geometry Components (p. 480).)

Renamed Geometry Properties The Properties that were renamed include the following:

- Area (now known as "Area : Cell")
- Nodal Area (now known as "Area : Nodal")
- Nodal X-Coordinate (now known as "Coordinate : Nodal : X")
- Nodal Y-Coordinate (now known as "Coordinate : Nodal : Y")
- Nodal Z-Coordinate (now known as "Coordinate : Nodal : Z")
- Displacement X (now known as "Displacement : X")
- Displacement Y (now known as "Displacement : Y")
- Displacement Z (now known as "Displacement : Z")
- Instantaneous Impact Power (now known as "Intensity : Impact")
- Instantaneous Shear Power (now known as "Intensity : Dissipation : Tangential")
- Instantaneous Normal Stress (now know as "Stress : Normal")
- Instantaneous Tangential Stress (now known as "Stress : Tangential")
- Instantaneous Nodal Forces X (now known as "Force : Nodal : X")
- Instantaneous Nodal Forces Y (now known as "Force : Nodal : Y")
- Instantaneous Nodal Forces Z (now known as "Force : Nodal : Z")

Removed Geometry Properties The Properties that were removed include the following:

- Mean Impact Power
- Mean Shear Power
- Mean Normal Stress
- Mean Tangential Stress
- Average Nodal Forces X
- Average Nodal Forces Y

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

- Average Nodal Forces Z

Tip:

Even though the above Properties are no longer directly computed in this version of Rocky, you may still use the Time Statistics feature to acquire this data if needed. (See also About Adding and Editing Time Statistics Properties (p. 1023).)

Removed Geometry Curves The Curves that were removed include the following:

- Impact Intensity X Belt Width
- Shear Intensity X Belt Width

Renamed Geometry Curves The Curves that were renamed include the following:

- Impact Intensity (now known as "Power : Impact")
- Shear Intensity (now known as "Power : Dissipation : Tangential")
- X-Force (now known as "Force : X")
- Y-Force (now known as "Force : Y")
- Z-Force (now known as "Force : Z")
- Normal Force (now known as "Force : Normal")
- Rotational Velocity X (now known as "Velocity : Rotational : X")
- Rotational Velocity Y (now known as "Velocity : Rotational : Y")
- Rotational Velocity Z (now known as "Velocity : Rotational : Z")
- Translational Velocity X (now known as "Velocity : Translational : X")
- Translational Velocity Y (now known as "Velocity : Translational : Y")
- Translational Velocity Z (now known as "Velocity : Translational : Z")

CHANGES AFFECTING MOTION FRAMES

(See also About Creating and Applying Motion Frames (p. 533).)

Renamed Motion Frame Curves The Curves that were renamed include the following:

- Rotational Velocity X (now known as "Angular Velocity X")
- Rotational Velocity Y (now known as "Angular Velocity Y")
- Rotational Velocity Z (now known as "Angular Velocity Z")

CHANGES AFFECTING PARTICLES

(See also About Adding and Editing Particle Sets (p. 602).)

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

Removed Particles Properties (Simulation-Wide) The Properties that were removed include the following:

- Maximum Normal Force Tip:You may get a more complete picture of this force by visualizing it from the **Contacts** entity. (See also About Contacts (p. 746).)
- Maximum Tangential Force Tip:You may get a more complete picture of this force by visualizing it from the **Contacts** entity. (See also About Contacts (p. 746).)
- Impact Specific Energy Tip:The Inter-particle Collision Statistics property called "Power : Impact" is nearly equivalent to this property. (See also About Inter-particle Collision Statistics (p. 331).)
- Shear Specific Energy Tip:The Inter-particle Collision Statistics property called "Power : Dissipation : Tangential" is nearly equivalent to this property. (See also About Inter-particle Collision Statistics (p. 331).)

CHANGES AFFECTING INTRA-PARTICLE COLLISION STATISTICS

(See also About Intra-Particle Collision Statistics (p. 324).)

Note:

This feature was known as *Particle Collision Statistics* in prior versions of Rocky. (See also I cannot find Particle Collision Statistics (p. 1330).)

Renamed Particle Set Properties (Per Set Only) The Properties that were renamed include the following:

- Nodal Area (now known as "Area : Nodal")
- Duration Mean (now known as "Duration : Mean")
- Duration Skewness (now known as "Duration : Skewness")
- Duration Std. Dev. (now known as "Duration : Std. Dev.")
- Intensity Normal (now known as "Intensity : Dissipation : Normal")
- Intensity Tangential (now known as "Intensity : Dissipation : Tangential")
- Stress Normal (now known as "Stress : Normal")
- Stress Tangential (now known as "Stress : Tangential")
- Velocity Normal Kurtosis (now known as "Velocity : Impact : Normal : Kurtosis")
- Velocity Normal Mean (now known as "Velocity : Impact : Normal : Mean")
- Velocity Normal Skewness (now known as "Velocity : Impact : Normal : Skewness")
- Velocity Normal Std. Dev.(now known as "Velocity : Impact : Normal : Std. Dev.")
- Velocity Tangential Kurtosis (now known as "Velocity : Impact : Tangential : Kurtosis")

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

- Velocity Tangential Mean (now known as "Velocity : Impact : Tangential : Mean")
- Velocity Tangential Skewness (now known as "Velocity : Impact : Tangential : Skewness")
- Velocity Tangential Std. Dev.(now known as "Velocity : Impact : Tangential : Std. Dev.")

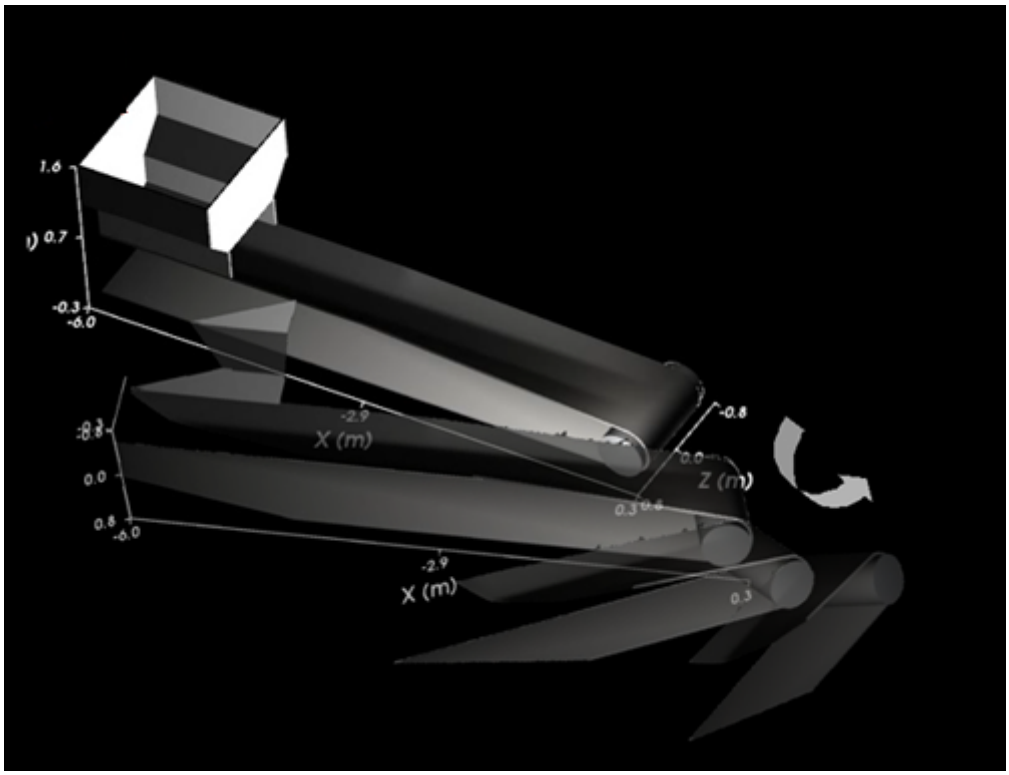
See Also:

- About Properties (p. 933)
- About Curves (p. 989)
- I cannot see the data or statistics (p. 1364)

8.4.13. Parts of my geometries and particles disappear when I rotate my 3D View

Occasionally, the objects you want to display in a 3D View window will be larger than the camera's current field of view. In these cases, geometries and/or particles can appear to be omitted from the 3D view when rotating the image (Figure 1).

Figure 8.30: Feed Conveyor with parts that disappear from the 3D View when rotated



If this happens, reset the camera's field of view by doing one of the following:

- Press the **R** key on your keyboard to **Fit** the active components into view
- Select a Camera Preset from the **Fit** list
- Close the current 3D View window and open a new one

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

See Also:

- About Using the Camera Visualization Toolbar (p. 1045)

8.4.14. I cannot display or plot a particular property

Most times, you can drag and drop a property from the Properties tab of an entity onto a 3D View, Particles Details, or Plot window, and it will display the data you selected. There are some exceptions, however, depending upon the type of property and/or type of window you use. See below for the most common exceptions.

Tensor Properties

Most properties in Rocky, such as mass or volume, are scalar quantities. This means they have a single value per particle (or per boundary triangle) and as such, can be displayed in a 3D View or be plotted.

A few properties in Rocky, such as Moment of Inertia and Stress Tensor (for Particles) are tensor quantities, which means that each value per particle (or per boundary triangle) is made up of nine separate components.

You can plot the components of the Stress Tensor in the Eulerian Statistics User Process (see also About Eulerian Statistics (p. 1226)), or you can export the components of a tensor quantity using scripts (see also About Creating and Using PrePost Scripts (p. 296)), but you cannot show each component in a 3D View or even plot them within Rocky.

Vector Properties

There are also a few vector properties, such as Translational Velocity (Particles) or Velocity (Fluids), which are made up of the x, y, and z directional components of the value. While the individual components of a vector property can be plotted or shown in a 3D View window, the vector property itself is limited to the 3D View only and cannot be plotted.

Space Plots

In Rocky, Space Plots are unique in that they require the data shown to be one-dimensional. Most Properties and Curves used in Rocky do not meet this criteria, so most of this data cannot be plotted in a Space Plot. There are a few specific conditions where Space Plots can still be useful for DEM analysis. (See also About Space Plots (p. 1118).)

See Also:

- About Properties (p. 933)
- About Space Plots (p. 1118)

8.4.15. My old script no longer works

This version of Rocky makes use of the Python 3 programming language. This means that scripts that worked in prior versions of Rocky, and which used an older version of Python (specifically, Python 2.7), may no longer work in this version.

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

To resolve this issue, try one or more of the following:

- Recreate your script using the latest version of Rocky. (See also Record a New Script (p. 301).)
- Edit your script in a Python editor and ensure that it meets the Python 3 criteria. (See also Edit a Saved Script Using the Python Programming Language (p. 303).)

See Also:

- About Creating and Using PrePost Scripts (p. 296)
- I still want to use my old macros (p. 1376)

8.4.16. I still want to use my old macros

In previous versions of Rocky, a script (p. 1394) that was recorded and played in the Rocky UI was known as a *macro*. Similar to scripts recorded and played in the **PrePost Script** panel today, these were PY files saved with a particular prefix; in this case, `macro_`.

In this version of Rocky, the *macro* concept has been replaced with the **PrePost Script** concept, but the underlying PY files are basically the same.

This means that if you have an older PY file saved with the `macro_` prefix, it can still be shown and played in the **PrePost Script** panel, just as a PY file saved with the `script_` prefix can be.

See Also:

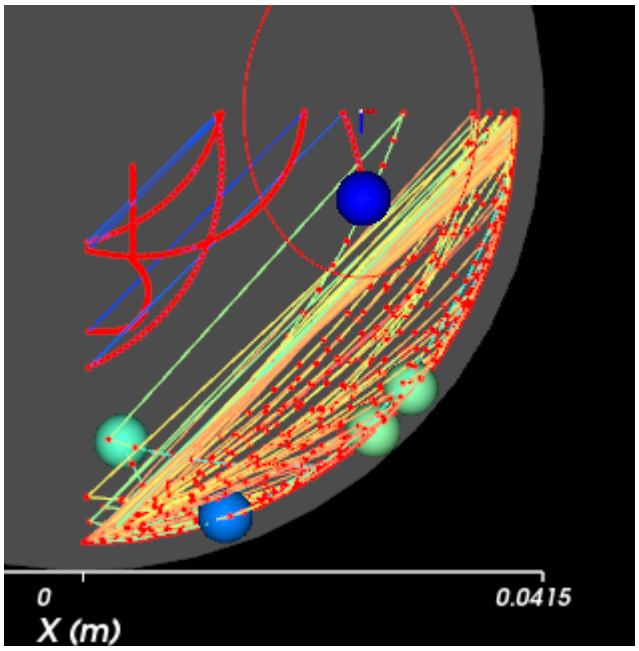
- About Creating and Using PrePost Scripts (p. 296)
- My old script no longer works (p. 1375)

8.4.17. My trajectory lines are outside my periodic domain

When using a Cylindrical Periodic Domain (see also About Domain Settings Parameters (p. 452)) and a Particles Trajectory User Process (see also About the Particles Trajectory User Process (p. 1209)), you might notice that the trajectory lines include particles that cross the periodic domain. This can generate results with additional (incorrect) trajectory lines (Figure 1).

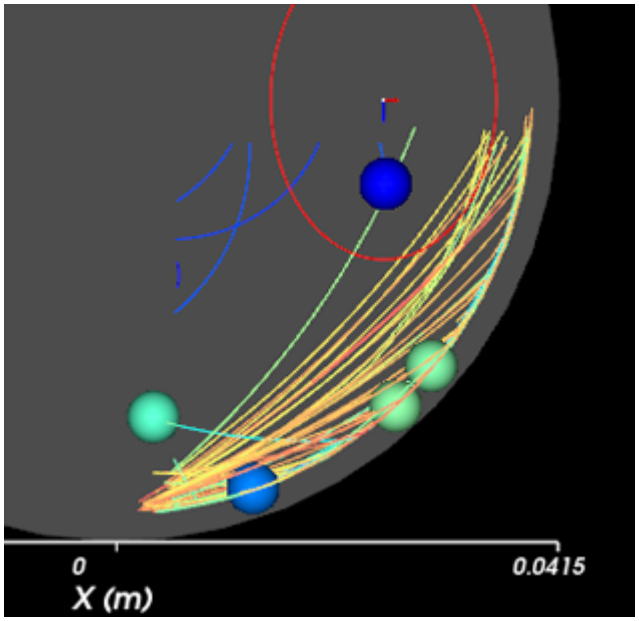
Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

Figure 8.31: Incorrect trajectory lines when using Cylindrical Periodic Domains



To avoid these errors, create a Cube or Cylinder User Process on your Particles Trajectory User Process, and make the dimensions a bit smaller than the dimensions of the Cylindrical Periodic Domain. This will serve to filter out the (incorrect) lines that are generated by the particles crossing the domain (Figure 2).

Figure 8.32: Cube User Processes added over the Particles Trajectory serves to filter out the incorrect lines



See Also:

- About Domain Settings Parameters (p. 452)
- About the Particles Trajectory User Process (p. 1209)

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

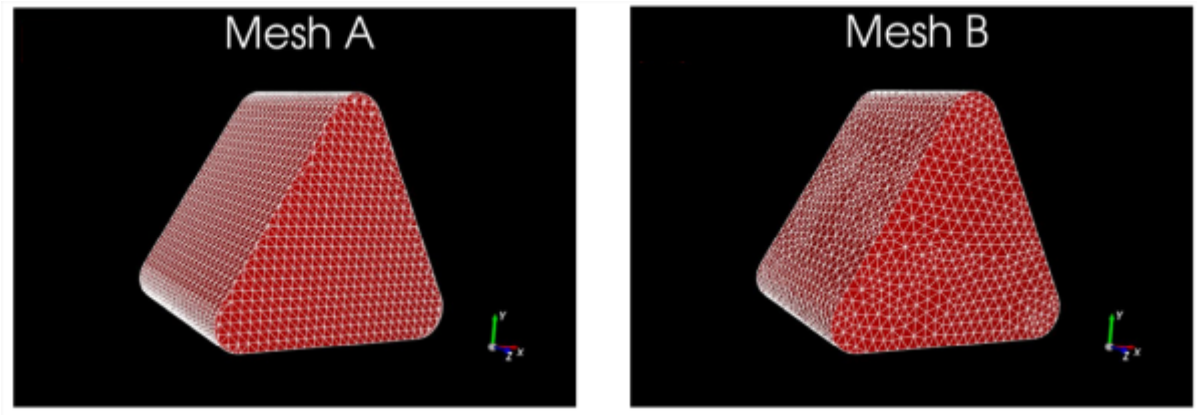
- About the Cube User Process (p. 1176)
- About the Cylinder User Process (p. 1181)

8.4.18. I have large discrepancies in my Intra-particle Collisions Statistics Frequency values

When you have chosen to collect Intra-particle Collisions Statistics (see also About Intra-Particle Collision Statistics (p. 324)) and are seeing very different Frequency values for similar particle shapes, this is likely due to the way the similar particle shapes are meshed, and in particular, how finely any large flat areas of the particle are meshed.

If the similar particle shapes are meshed in different ways, as shown in Figure 1, you might get slightly different results due to the way that Rocky categorizes and treats the geometries as convex and concave shapes respectively. (See also About Defining and Importing Custom Particle Shapes (p. 670).)

Figure 8.33: Mesh A is treated by Rocky as a convex shape; Mesh B is treated as a concave shape



However, if the similar particles also include large flat portions discretized with a large number of triangles, larger discrepancies in Frequency values might be seen in the following circumstances:

- The overlaps are large (e.g., in the order of the triangles size).
- Side-to-side collisions are frequent.

Discrepancies in Frequency arise in these cases because for a concave particle shape, a collision on the flat side will involve *all* the triangles that make up the flat side; for a convex particle, a collision on the flat side will likely involve only *one* triangle.

Refer to the **DEM Technical Manual** for more information. (From the Rocky Help menu, point to Manuals, and then click DEM Technical Manual.)

See Also:

- About Intra-Particle Collision Statistics (p. 324)
- About Defining and Importing Custom Particle Shapes (p. 670)
- About Properties (p. 933)

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

8.4.19. I cannot find the directional properties for Solid Implicit Force

In versions of Rocky prior to 2022 R1, after processing your 2-Way Fluent coupling cases there were made available the following vector and associated directional components to the implicit source term property:

- Solid Implicit Force
- Solid Implicit Force X
- Solid Implicit Force Y
- Solid Implicit Force Z

Because the directional components were all the same value, and did not contain any directional information, the vector and associated directional components were replaced by a single property called **Solid Implicit Force** in 2022 R1, which better reflected the data while reducing the overall storage requirements.

See Also:

- About Using the 2-Way Fluent Method (p. 795)
- About Properties (p. 933)
- Troubleshooting Simulation Analysis Problems (p. 1363)

8.5. Troubleshooting Rocky Licensing and File Problems

What kind of problem are you having?

- Troubleshooting Common Licensing or File Errors (p. 1379)
- Other licensing or file issues:
  - Rocky says I have lost the connection with the license server (p. 1384)
  - My Rocky license will not let me open a project from the File directory (p. 1385)
  - I cannot find the import project functionality (p. 1387)
  - I cannot open my older simulation file in the latest version of Rocky (p. 1387)
  - I made a copy of my older simulation project but Rocky will not open it (p. 1388)

See Also:

- Troubleshooting (p. 1315)

8.5.1. Troubleshooting Common Licensing or File Errors

What kind of problem are you having?

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

- Rocky says I have "No Solver License" when I process my simulation but my license status is "OK" (p. 1380)
- My Rocky license Status says "No license for feature" (p. 1380)
- I get a "Links removed" message when I save my project for restart purposes (p. 1381)
- I get a "No Solver license available" message when I run the Rocky Scheduler but the status says "License Ok" (p. 1340)
- Rocky says my project file is "locked" (p. 1382)
- I reinstalled Rocky but I am still seeing errors (p. 1384)

See Also:

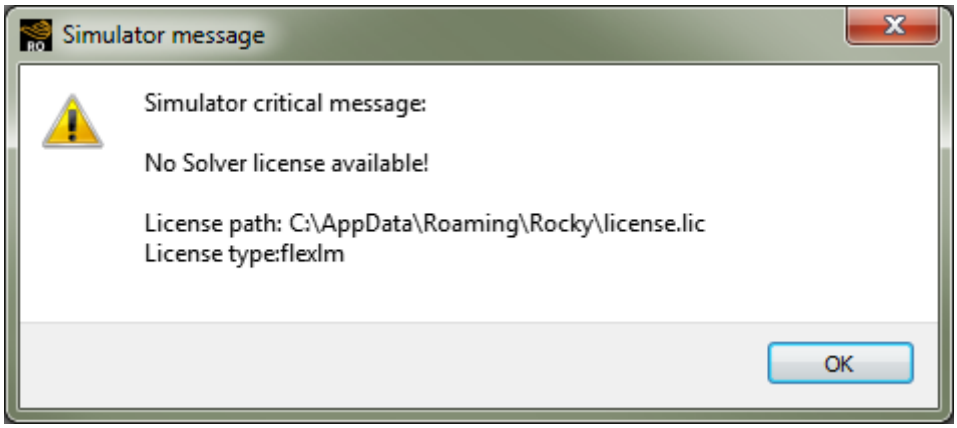
- Troubleshooting Rocky Licensing and File Problems (p. 1379)

Rocky says I have "No Solver License" when I process my simulation but my license status is "OK"

If after you start processing your simulation, you get a Rocky message saying you have "No Solver license available" (see Figure 1 below), but your license Status says **OK** (see also Verify Your Rocky License Status (p. 1295)), then there might be a hidden Rocky process running in the background that is using your license. Even restarting Rocky might not clear all hidden processes in some cases.

To solve this issue, try restarting your computer. This should clear the hidden processes making use of your license.

Figure 8.34: Simulator message



See Also:

- About Starting a Simulation (p. 908)

My Rocky license Status says "No license for feature"

If attempting the above does not solve the problem, Submit a Support Request (p. 1293) for additional assistance.

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

If when you Verify Your Rocky License Status (p. 1295), you see a message that indicates an issue with your license, Submit a Support Request (p. 1293) for additional assistance.

See Also:

- About Processing Multiple Simulations In Succession (p. 920)
- Appendix G: Perform Tasks from the Command Line or Journal (p. 1308)

I get a "Links removed" message when I save my project for restart purposes

With a few exceptions (see also I cannot enter an input variable or mathematical function into a text field (p. 1334)), most Rocky fields enable you to enter variables and/or mathematical functions instead of direct, numerical values. When you save a project with variables or functions for restart purposes, all but a few of these fields will retain the link between the value shown and the parametric expression that was entered.

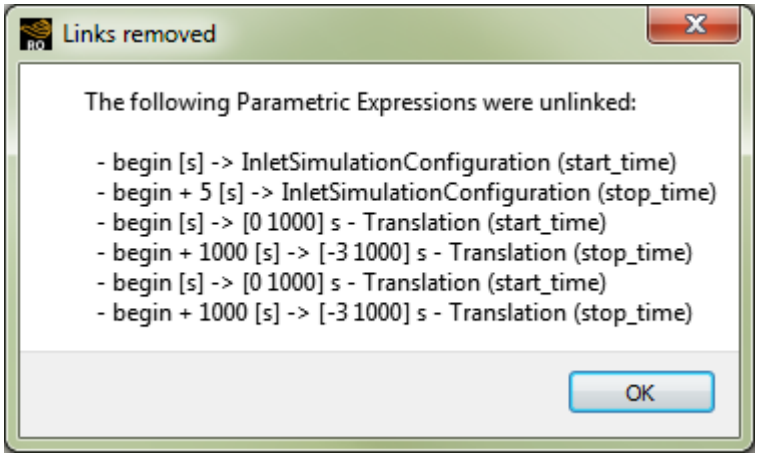
The few Rocky fields that will *not* retain this link after being saved for restart include:

- Both the **Start Time** and **Stop Time** parameters for Motion Frames. (See also About Creating and Applying Motion Frames (p. 533).)
- Both the **Start Time** and **Stop Time** parameters for continuous injection inputs. (See also About Adding and Editing Particle Inputs (p. 708).)
- Both the **Enable Time** and **Disable Time** parameters for Imported Geometry components.

This means that in the project copy you have saved for restart, instead of being defined by the parametric expression you have entered, these few fields will instead be defined only by the resulting value adjusted due to restart. For example, if the resulting value was 5 seconds and the restart project was created at time 2 seconds on the original project, the new value in the unlinked field will be 3 seconds.

When you initially save the project copy, Rocky will remind you about which fields this affects via a "Links removed" message similar to the one in Figure 1.

Figure 8.35: Example Links removed message



Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

If the field is enabled in the project copy, you may use the resulting field values as is or you may replace them with new parametric variables.

See Also:

- Enter Input Variables or Mathematical Functions as Parameter Values (p. 372)
- Copy of a Partially Processed Simulation for Restart Purposes (p. 855)
- I cannot enter an input variable or mathematical function into a text field (p. 1334)

I get a "No Solver license available" message when I run the Rocky Scheduler but the status says "License Ok"

Most users of Rocky will have single-instance licenses, which means that only one instance of the Rocky solver can be running at any one time. For these users, it is possible to use the Rocky Scheduler program to run multiple simulations in series, or one-by-one, but not in parallel or simultaneously.

If you have a single-instance license and attempt to start processing more than one simulation at a time, you will get a "No Solver license available" message for all but the first simulation, as it will be using the single license that was available. Once that first simulation is done processing, the license will be freed up to be used by the next simulation in the queue, and so on.

If you have a need to process multiple simulations simultaneously, please Submit a Support Request (p. 1293) or contact your Rocky representative to ask about upgrading to an unnumbered Rocky license.

See Also:

- About Processing Multiple Simulations In Succession (p. 920)

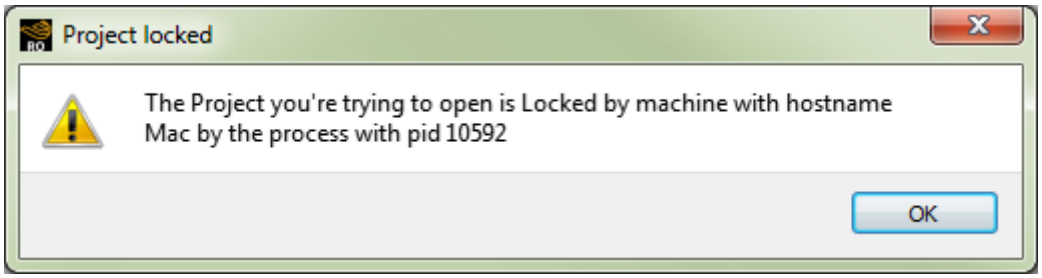
Rocky says my project file is "locked"

Rocky automatically "locks" a project file to the instance of Rocky that has opened it, which helps prevent yourself and other users on your network from accidentally opening up a duplicate instance of the project and making changes that cannot be reconciled with the original project file. Rocky does this by adding a `.lock` file to the project directory when a project is opened. This lock file contains information about which system and Rocky instance is allowed to edit and save the active project. When the project is closed, Rocky automatically removes the lock file.

If you attempt to open a Rocky project file and are told by Rocky that it is locked via a message similar to the one in Figure 1, then it is likely that you or another user in your network has that project open already.

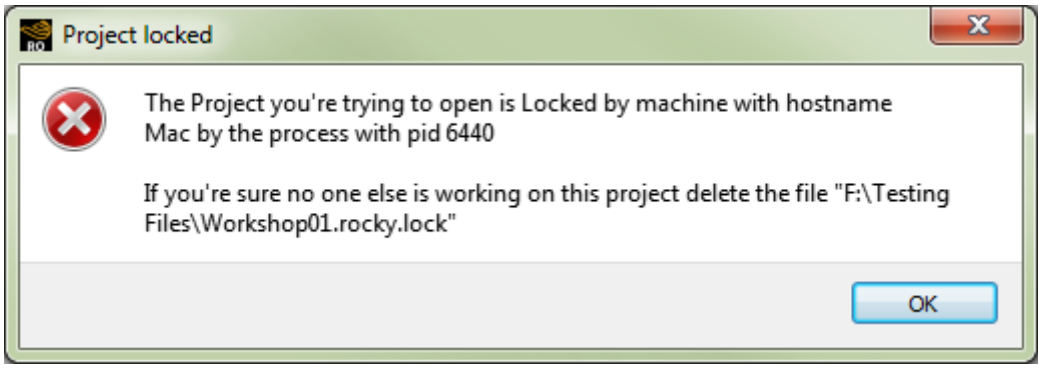
Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

Figure 8.36: Example message you might see if you try opening a project that has been locked



You may still open a locked project if you wish by clicking **OK** on the message, but know that this is a "read-only" copy: any changes you make to the settings will not be savable as long as the project remains locked. If you do attempt to save the changes to this project while it is still locked, you may get a slightly different message from Rocky reminding you about its current status (Figure 2).

Figure 8.37: Example message you might see if you try saving a project that has been locked



In order to open a fully editable version of the project yourself, the other instance of the project file must first be closed.

There are some circumstances under which the `.lock` file might still be present for a project when it should have been removed. For example when the project has closed incorrectly, such as during a power outage or a forceful system reboot. In those cases, the process identifier used by Rocky to lock the file might have been assigned to another process, which might cause the program to believe Rocky is still being used when it isn't.

In these cases, if you are certain no one else is working on the file, you should manually delete the `.lock` file from the directory.

Notes:

- Nothing prevents other users on your network from deleting the `.lock` file and taking ownership of the project you are working on while you are working a project. If this happens, you may suddenly be locked out of your own project and be prevented from saving changes until the other user closes their copy of the project and it becomes locked to your instance of Rocky again.
- Conversely, if you are reviewing a locked project file, you may suddenly find yourself able to save changes to it if during your review, the user who had originally opened the project file closed it. In this case, it is important to be aware that you can make changes to the project that override the changes the other user had just made.

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

See Also:

- Open a Simulation Project (p. 411)

I reinstalled Rocky but I am still seeing errors

If you are experiencing unexplained problems in the Rocky product—such as no content in the **Offline User Manual**, frequent shutdowns, or repeated errors for seemingly no reason—and those issues do not go away even after you uninstall and reinstall Rocky, then you might require a clean install.

Rocky uses several folders on your machine to store the files it needs. (See also the *Common Folder Locations Used in Rocky* section of the File Types and Folders (p. 129) topic.) Sometimes, those files can become corrupted, causing problems within the Rocky product itself.

Because these files can persist on your machine even after you uninstall and reinstall Rocky, there may be times when you need to fully clear these files prior to a reinstall.

To ensure a clean install of Rocky:

Note:

This procedure works only on Windows machines.

Warning: Following this procedure may result in a loss of your previously created Custom Curves and/or Custom Properties, your license configuration, your display and plot settings, and any other global preferences you might have defined.

1. Follow the instructions to Remove Rocky (p. 1296).
2. Rename and/or archive your `../Documents/Rocky` folder. Tip:Renaming and/or archiving this folder—rather than deleting it outright—ensures that you can still access these files later if you need them.
3. Rename and/or archive your `C:Users/USER/AppData/Roaming/Rocky` folder.
4. Delete the `Rocky` folder in your `C:Users/USER/AppData/Local` folder.
5. Restart your machine.
6. Reinstall Rocky.

See Also:

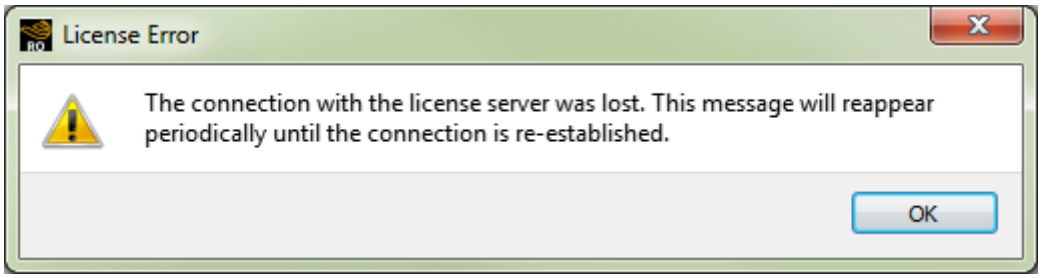
- File Types and Folders (p. 129)
- Troubleshooting Rocky Licensing and File Problems (p. 1379)

8.5.2. Rocky says I have lost the connection with the license server

At any point using the program, Rocky might present you with a "License Error" stating that you have lost your connection to the license server (Figure 1).

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

Figure 8.38: License error announcing lost connection



For security purposes, Rocky periodically attempts to communicate with the license server and if it cannot reach it- likely due to server errors or internet connection problems-it will present this message. Rocky will keep trying to reach the server until it is successful. In the meantime, you should still be able to use Rocky.

If you find that the message is appearing frequently and interferes with your ability to use the program, check your internet connection, and try restarting your computer. If your connection is good and re-starting your computer did not help, then Submit a Support Request (p. 1293) to request assistance.

See Also:

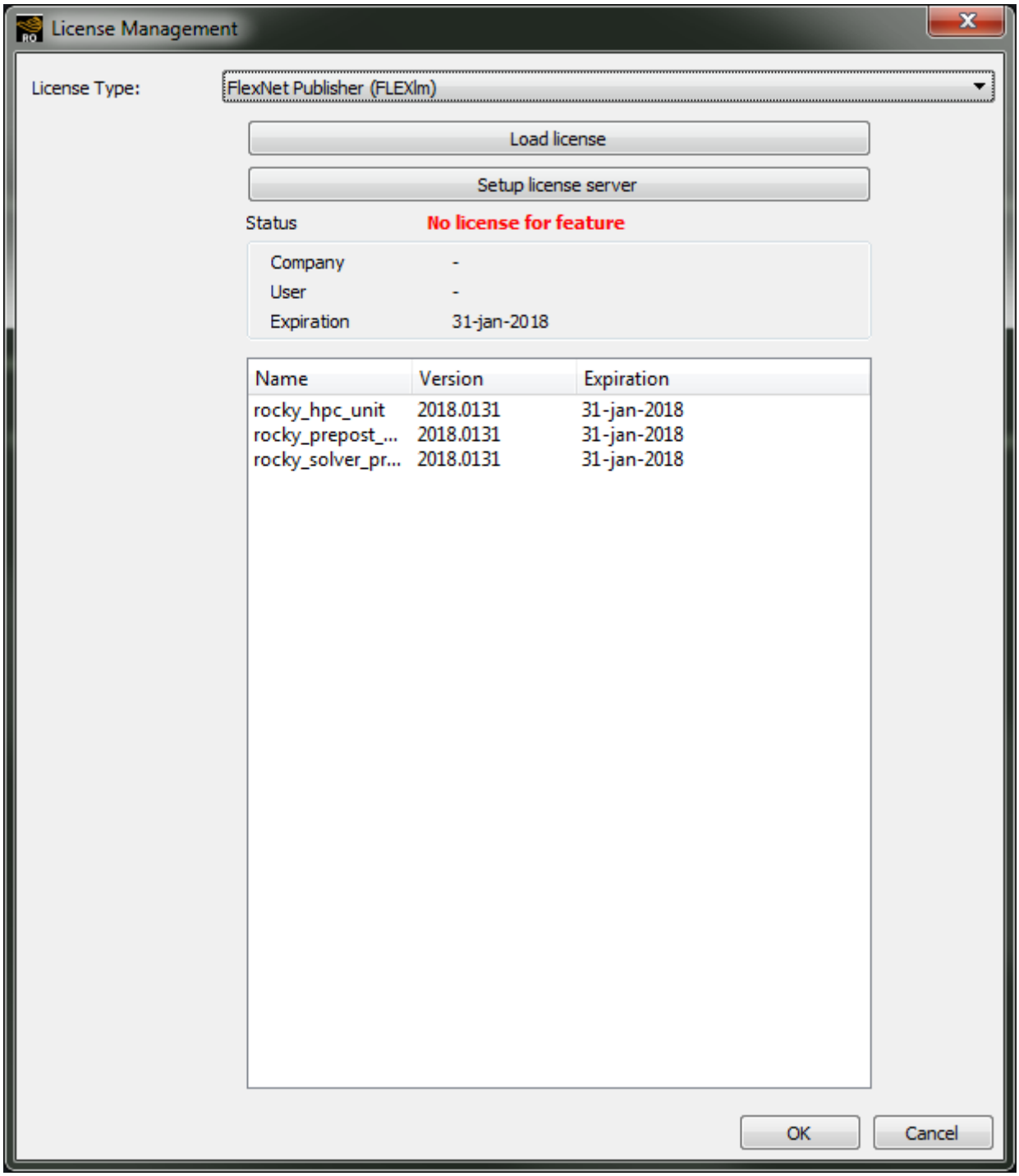
- Getting Help and Support (p. 1285)

8.5.3. My Rocky license will not let me open a project from the File directory

If you already have the Rocky program open and then attempt to use the File directory to open a Rocky project instead of the Rocky File menu, you might instead see the License Management screen with the Status of "No license for feature" (Figure 1).

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

Figure 8.39: Example - License Management dialog showing a Status of No license for feature



If you have a single-instance license of Rocky and the Rocky program is already open, then attempting to open a project from the File directory causes Rocky to want to open a second instance, which a single-instance license might not allow.

To fix this, try any or all of the following:

- Use the File directory to open a Rocky project only when Rocky is not already open.

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

- To open another project when Rocky is already open, use only the **File** | **Open Project** menu item from within Rocky. (See also Open an Existing v4 or Later Simulation Project (p. 413).)

If attempting the above does not solve the problem, Submit a Support Request (p. 1293) for additional assistance.

See Also:

- About Processing Multiple Simulations In Succession (p. 920)
- Appendix G: Perform Tasks from the Command Line or Journal (p. 1308)

8.5.4. I cannot find the import project functionality

As of Rocky version 4.2, it is no longer possible to import older v2 or v3 simulation projects directly into Rocky. To learn other ways to use your older simulation projects in newer versions of Rocky, see I cannot open my older simulation file in the latest version of Rocky (p. 1387).

See Also:

- Open a Simulation Project (p. 411)

8.5.5. I cannot open my older simulation file in the latest version of Rocky

Newer versions of Rocky are not specifically designed to open simulations generated in older versions. As such, compatibility with these versions is not guaranteed.

However, you can use the methods below, if you want to try opening an older version:

Open a simulation generated in v3: This will open the setup, simulation results, and available post-processing information from the older simulation project in the latest version of Rocky but will limit the features to only those that were available in Rocky v3. (See also Open an Existing v3 Simulation Project (p. 412).) Important Opening an older simulation file in a newer version of Rocky will prevent you from being able to open the original simulation files in older versions of Rocky. If you need to retain the ability of opening the original simulation in an older version of Rocky, it is best to make a copy of the simulation file and folder and then open or import the copy. (See also I made a copy of my older simulation project but Rocky will not open it (p. 1388).)

Open a copy of the older simulation project in Rocky: You can retain your ability to open the original file in an older version of Rocky by first making a copy of the older simulation project file and related folder, and then opening the copy in the newer version of Rocky. There are several ways you can make this copy, as follows:

- If you have access to the version of Rocky that the older file was created in, you can either Save a Copy of Only the Simulation Settings; Not the Simulation Results (p. 854) (fastest process) or Save an Exact Copy of the Simulation Project; Including the Simulation Results (p. 853) (slowest process) directly from within the older version of Rocky.
- If you do not have access to the version of Rocky that the older file was created in, you can make a copy of the original project from your computer's directory system. Be sure that you copy both the simulation file (.rocky30) and its related folder, which will have the same prefix as the project file. For Example: If your project is called "my_project" then you should make a copy of both the

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

"my_project.rocky30" file and the "my_project_file_dir.rocky30" folder that is next to it. Note: Be sure to name the copied files in exactly the format that Rocky expects. If you are copying the original files from within a Windows directory, this can involve an extra step. (See also I made a copy of my older simulation project but Rocky will not open it (p. 1388).)

See Also:

- Save a Simulation Project (p. 852)

8.5.6. I made a copy of my older simulation project but Rocky will not open it

If you need to open in the latest version of Rocky a simulation project generated in an older version of Rocky, one of the ways you can retain your ability to open the original simulation in the older version of Rocky is to create a copy of the original simulation file and folder and open the copy in the newer version of Rocky.

Opening the copy of your simulation project will work only if the copy is named in exactly the format that Rocky expects it to be. For example, a Rocky v3 project called "my_project" will be formatted as follows:

- *my_project.rocky30* (simulation project file)
- *my_project_file_dir.rocky30* (simulation project folder)

When making copies in Windows-based operating systems, it is especially important to pay attention to this naming. A copy and paste in a Windows directory folder will automatically append the words "- Copy" to the end of the duplicate file and folder name. For example, making a copy of the "my_project" file and folder above would generate the following two items:

- *my_project - Copy.rocky30* (simulation project file copy)
- *my_project_file_dir.rocky30 - Copy* (simulation project folder copy)

Opening the new *my_project - Copy.rocky30* file in the latest version of Rocky will result in errors because Rocky will be looking for a companion folder called *my_project - Copy_file_dir.rocky30* that doesn't exist. It is therefore very important that when copying your project file and folders-especially in Windows-based operating systems-you rename them in a way that Rocky can find them.

See Also:

- I cannot open my older simulation file in the latest version of Rocky (p. 1387)

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

Glossary

Use the definitions below to help you understand the various terminology used within the Rocky user interface (UI) and user assistance content (this document).

A to M

airflow	Short for "1-Way LBM", and previously known as "Lattice Boltzmann Air Flow", this method uses coupled gas dynamics to simulate how air, dust, or other gas flows around the particles and boundaries in your chute design. This kind of CFD is done entirely within Rocky and is not coupled with Ansys.
Assembly	When referring to particle shapes, these are made by combining the shapes of two or more rigid, *Particle sets* (p. 1393) to form a single, unique particle.
block	One of many cube-like segments that were made by applying Eulerian Statistics to a Cube or Cylinder User Process.
boundary (geometry)	When referring to geometries, boundaries are the physical barriers, such as chute walls and conveyor belts, that make up the materials handling design. Boundaries are what particles bounce off of during a simulation to determine flow and wear of the various components. There are two default boundaries included within Rocky: receiving conveyors and feed conveyors. All other boundaries, such as chutes, mills, and hoppers, are imported into Rocky.
boundary (simulation)	When referring to the simulation as a whole, boundaries are the coordinate limits of the objects currently active in the simulation domain. These boundaries are used by Rocky to make calculations.
breakage (discrete)	(See *Discrete Breakage* (p. 1390).)
breakage (instantaneous)	(See *Instantaneous Breakage* (p. 1392).)
CAD software	Computer-Aided Design (CAD) programs are drafting software used by engineers and designers to create 2D or 3D drawings of various objects and components. Ansys SpaceClaim is one example of this type of software.
cell (airflow)	A cube-like segment into which an *airflow* (p. 1389) boundary is divided. Simulations are calculated based upon how particles interact with these segments. Therefore, the more cells an air flow boundary is divided into, the more accurate the simulation results will be.
cell (simulation)	Refers to the individual component-either a *triangle* (p. 1395), *block* (p. 1389), *node* (p. 1393) or *air flow cell* (p. 1389)-that can be analyzed by the Cell Inspector.

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

CFD software	Computational Fluid Dynamics (CFD) is a type of simulation software that solves and analyzes problems involving the flow of fluid. Ansys Fluent is an example of this type of software.
collision	An event in time when a particle touches something else (*boundary* (p. 1389) or other particle), and which includes one or more *contacts* (p. 1390).
concave	When referring to particle shapes, these are closed 3D whole or perforated shapes of full interior volume that contain at least some faces that form dents or hollows.
contact	A specific location on a *boundary* (p. 1389) or other particle that has experienced a *collision* (p. 1390).
convex	When referring to particle shapes, these are closed 3D whole shapes of full interior volume that contain no faces that form dents or hollows.
CPU processing	Refers to the ability of Rocky to made use of the Central Processing Unit (CPU) within a computer to perform calculations and render images and animations.
Curve	A data set applying to a simulation entity that includes one-dimensional recorded values measured across time or other properties. Curves are available for conveyors, imported geometries, Particles, and the Solver. As with Properties, individual User Processes linked to any entities with Curve properties will also display the same properties on their own Curves tabs.
custom particle	A unique *shaped particle* (p. 1395) saved as an .stl file (Custom *Shell* (p. 1395) and Custom *Solid* (p. 1395) Polyhedron), or a text or spreadsheet file (Custom *Fiber* (p. 1391)) that is imported into Rocky. The .stl shapes can be designed in a *CAD program* (p. 1389) (Custom Shell and Custom Polyhedron), or can be generated from a 3D scan of an actual object (Custom Polyhedron).
default conveyor	One of two adjustable conveyor geometry templates included by default in Rocky: a feed conveyor and a receiving conveyor.
Discrete Breakage	Commonly referred to as "joint breakage" or "discrete grain breakage (DGB)", this refers to the type of breakage that happens between the individual *joints* (p. 1392) of a flexible particle that is composed of multiple elements.
Element	A component that together with other (roughly) equal-sized Elements, comprises a particle shape that has been *meshed* (p. 1392), which depending upon the shape type, can enable features such as flexibility and/or breakage.
embedded Module	A *Module* (p. 1392) that comes by default installed with the Rocky product. This kind of Module is documented in the User Manual (this document) and Technical manuals.

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

exclusive setting	Refers to a particular model or setting in the Rocky UI that when affected by an enabled *Module* (p. 1392), has the effect of disabling Rocky's default options for that particular model or setting, and then replacing them with the name of the Module.
external Module	A *Module* (p. 1392) that is installed separately from the Rocky product via a ZIP file. Because it is separate from the product, its documentation (if available) is provided separately from the Rocky product, and can usually be accessed from the **Data Editors** panel by clicking the (?) Help button from the Module's main tab.
FEA software	Finite Element Analysis (FEA) refers to the act of studying or analyzing a phenomenon that has been modeled with the Finite Element Method *FEM* (p. 1391). An example of FEA software for structural analysis is Ansys Mechanical.
feed conveyor	A default conveyor template included within Rocky that provides an adjustable feeder box, the top of which acts as the inlet from which particles are dropped, and an attached conveyor and head pulley.
FEM	Finite Element Method (FEM) refers to a method for numerically solving various engineering and mathematical models, such as those commonly applied within structural and electromagnetic simulation software. The tools that make use of this method are often referred to as *FEA software* (p. 1391).
Fiber	A category of particle shapes that are made by stringing together linear *Segments* (p. 1394)-each with its own thickness- to form strand-like 3D particles. Rocky comes with a **Straight Fiber** by default, but other **Custom Fiber** shapes can be imported with a text or spreadsheet file.
free boundary	A geometry imported into Rocky that has a **Motion Frame** with free body motion enabled. Free body motion allows the *boundary's* (p. 1389) movement to be affected by gravity, any additional (prescribed) forces/moments that you define, and by the particles with which it comes into contact. (Note that boundaries are not affected by other boundaries.)
frozen Fiber	Refers to the ability to define for a Custom *Fiber* (p. 1391) one or more *Segments* (p. 1394) that are "frozen". When a frozen Fiber is composed of Multiple *Elements* (p. 1390) and combined with Custom Inputs, this allows the frozen portion of the multi-element Fiber to "stick" to a certain location and have only the unfrozen portions of the fiber respond to interactions from other particles and boundaries in the simulation.
geometry	*CAD* (p. 1389) files that make up the objects used in a simulation, including conveyors, boundaries, and custom particle shapes.

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

GPU processing
: Refers to the ability of Rocky to make use of a Graphics Processing Unit (GPU) computer chip to perform rapid calculations and quickly render images and animations on screen.

imported geometry
: A custom boundary, such as a chute, mill, or hopper, that is imported into Rocky from a *CAD program* (p. 1389).

Info
: A **Data Editors** tab that contains the domain properties for an *entity* (p. 1395), such as the topology and geometry coordinate limits, that provide context as to how the simulation data was recorded. These **Info** properties are made available for conveyors and imported geometries, Particles, 1-Way LBM, 1-Way Fluent, and individual User Processes.

inlet
: A default geometry included with Rocky that provides a rectangular or circular plane from which particles can be released into the simulation. Inlets can be located anywhere within your simulation and can be of any rectangular or circular size, and of any angular orientation.

Instantaneous Breakage
: Commonly referred to as "Instantaneous Fragmentation (IF)", this refers to the type of breakage that causes rigid, sharp-cornered particles to fragment into smaller pieces.

joint
: The connection or "glue" between two individual *Elements* (p. 1390) of a multi-element (*meshed* (p. 1392)) particle.

macro
: (Term no longer used. See *script* (p. 1394) instead.)

Meshed
: A model in Rocky that enables certain Particles to be divided (or tessellated) into (roughly) equal-sized *Elements* (p. 1390), which can then be used to enable features such as flexibility and *Discrete Breakage* (p. 1390).

Module
: Discrete features or functionality provided via code that is separate from the Rocky product. Modules can either be *embedded* (p. 1390) and installed with the Rocky product, or *external* (p. 1391) and installed later via a ZIP file.

multi-branched
: Refers to a *Fiber* (p. 1391) or *Shell* (p. 1395) particle shape that includes one or more bifurcating points where a *Segment* (p. 1394) or surface divides into a separate branch or fork.

multiphase
: With 1-Way and 2-Way Fluent coupled simulations, this refers to the use of two or more CFD fluid phases with one DEM particle phase. (Sometimes also referred to as "multi-phase".)

N to Z

neighbor particle
: A particle near enough to another particle or boundary to be considered for a *collision* (p. 1390) event.

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

neighbor triangle	A boundary *triangle* (p. 1395) near enough to a particle to be considered for a *collision* (p. 1390) event.
node	The point at which a collection of boundary *triangles* (p. 1395) intersect.
non-exclusive setting	Refers to a particular model or setting in the Rocky UI that when affected by an enabled *Module* (p. 1392), has the effect of adding in additional settings or options for the Module, but does not override any default options or settings.
output file	Previously referred to as "timestep files" and also known as RHS, RHC, or RHM files, these are the individual particle and boundary (RHS), contact (RHC), and motion (RHM) data files that together make up an entire simulation. During processing, one set of output files are saved per simulation output frequency. Output files are numbered sequentially starting from zero but are not linked to the point in time they represent. For example, when no Time Filter is applied, an output file of 0 (zero) corresponds to 0 (zero) seconds in the simulation. When a Time Filter of 2 s to 5 s is applied, an output file of 0 (zero) corresponds to whatever the beginning output time of that range is, or 2 s.
output time	Refers to a time in the simulation when an *output file* (p. 1393) has been saved. You can select an output time from the **Time** toolbar.
panel	A Rocky UI element containing parameters for setting up, displaying, running, and analyzing a simulation. Panels are docked by default but can be made to float over the rest of the UI.
part	When referring to particle shapes, this is an individual component of the whole *Assembly* (p. 1389) shape.
particle	Shapes used to represent an individual piece of matter-such as a rock, a kernel of grain, or a piece of hay-as it flows through bulk materials handling equipment. Simulated particles can be of various sizes, shapes, weights, and densities to more accurately reflect real matter.
particle fragment	Refers to an individual broken piece of a once whole *particle* (p. 1393). Occurs only in simulations with breakage modeling. Note that a fragment can itself be broken into more fragments.
particle group	In this version of Rocky, this is identical to *Particle set* (p. 1393).
Particle set	A collection that includes the particle shape type combined with other defining characteristics, including size distribution, rolling resistance, rotation, orientation, flexibility, and breakage capabilities.
plot	Refers to one of several different types of graphs in Rocky, such as a Time Plot or Cross Plot, that shows the relationship between two or more *Properties* (p. 1394) or *Curves* (p. 1390).

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

PrePost Script	(See *script* (p. 1394).)
project	Refers to all information and settings saved within a .rocky file, including setup information, simulation results, and post-processing analyses such as animations, graphs, and views.
Property	A post-processing data set that includes recorded values for each individual structural element at each individual *output time* (p. 1393) during a simulation. Properties are available for simulation entities including conveyors and imported geometries, Particles, Contacts, 1-Way LBM, 1-Way Fluent, and individual User Processes.
PSD	Short for "Particle Size Distribution", which defines the amount of particles present in a sample according to size.
receiving conveyor	A default conveyor template included with Rocky that provides an adjustable conveyor used to capture the flow off a chute or feed conveyor.
remeshed	Refers to a Multiple Element (*meshed* (p. 1392)) particle that Rocky has divided (or tessellated) into a different amount of *Elements* (p. 1390) than was defined for the shape originally.
rendered geometry	Refers to an imported or default *geometry* (p. 1391) component that has been saved, displayed, and/or modified-via wear or parameterization-within the confines of the Rocky project. These actions can render the original geometry differently and can therefore effect the results that are exported.
rendered shape	Refers to an imported or default *particle* (p. 1393) shape that has been saved, displayed, and/or modified-via internal algorithms or parameterization-within the confines of the Rocky project. These actions can render the original shape differently and can therefore effect the results that are exported.
rounded-corner shape	Refers specifically to a *shaped particle* (p. 1395) that is composed of rounded corners (Sphero-Cylinders, Sphero-Polygons, Sphero-Polyhedrons).
script	Short for *PrePost Script*, which is a set of repeatable project setup and/or data analysis tasks accomplished in the Rocky UI that was either 1) recorded click-by-click using only the **PrePost Script** panel, or 2) written outside of Rocky using Python code that references Rocky's PrePost Scripting functionality. Once recorded or written, can be played from the **PrePost Script** panel, added to the **PrePost Scripts** list under the **Solver** entity, and/or be launched from the command line.
Segment	A single, one-dimensional link that forms the basis of a *Fiber* (p. 1391) particle. One Segment is equal to a **Straight Fiber** shape. Multiple Segments, defined by the node locations within the text or spreadsheet file that defines a **Custom Fiber**, can be strung together to create various other 3D shapes.

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

shaped particle	Commonly referred to as a truly "non-round" or "non-spherical" particle, this is a way of describing any realistic particle shape that is not a perfect Sphere, nor made of "glued" Spheres. This category includes shapes with *sharp corners* (p. 1395) and *rounded corners* (p. 1394), *Fiber* (p. 1391) and *Shell* (p. 1395) shapes, and any other (non-Sphere) shapes in between.
sharp-cornered shape	Refers specifically to a *shaped* (p. 1395) particle that is made up of flat planes and sharp corners (Polyhedrons, Briquettes, Faceted Cylinders, non-concave Custom Polyhedrons).
Shell	A category of particle shapes that are made by joining two-dimensional, open surfaces of a definable thickness to create an empty, 3D container or sheet-like object. Because they contain no interior volume, the mass of a Shell particle is much lower than a similarly shaped *Solid* (p. 1395) particle, as the mass is confined only to the thickness of the container's surfaces. No default Shell particles come with Rocky, but you can add your own **Custom Shell** shapes by importing an .stl file.
simulation entity	A whole Rocky component for which settings are made and Properties and/or Curve properties are provided. These include Particles, 1-Way LBM, 1-Way Fluent, Solver, individual geometry components, individual Motion Frames, and any individual User Processes linked to these entities.
single phase	With 1-Way and 2-Way Fluent coupled simulations, this refers to the use of only one CFD fluid phase along with one DEM particle phase.
Solid	A category of particle shapes that are closed three-dimensional objects of full interior volume. Seven Solid shapes are provided by default with Rocky, and you can also import your own convex or concave **Custom Polyhedron** shape from an .stl file.
spherical particle	Commonly referred to as a "round" particle, this is a way of describing a particle shape that is a perfect Sphere.
structural element	The smallest individual component of a Rocky *entity* (p. 1395) upon which calculations are made. These include individual particles or *Elements* (p. 1390), 1-Way LBM *cells* (p. 1389), and boundary *triangles* (p. 1395). Many structural components make up an entire simulation entity.
time step	In simulation software like Rocky, a time step is the incremental change in time for which the calculations are being solved. Can also refer to an *output time* (p. 1393), which is displayed and selected on the **Time** toolbar.
timestep files	(Term no longer used. See *output file* (p. 1393) instead.)
triangle	A three-sided segment into which a geometry *boundary* (p. 1389) is divided. Simulations are calculated based upon how particles interact with these segments. Therefore, the more triangles a boundary

	has (i.e., the lower the triangle size) the more accurate the simulation results will be.
upscaling	Refers to the way data and statistics from the individual *Elements* (p. 1390) of a Multiple Element (*meshed* (p. 1392)) particle are considered when displaying the resulting properties or curves after processing. Even though the Rocky solver always calculates the underlying properties per Element, an upscaled particle (default setting) will result in properties or curves that affect the whole particle; a particle with upscaling turned off will result in properties or curves that affect each individual Element making up the particle.
window	Method by which the physical components or the graphical results of the simulation are viewed or previewed. Includes 3D View, plots, histograms, Particles Details, and Motion Preview windows.
Workspace	The area in the center of the Rocky UI where collections of windows are grouped together on individual Workspace tabs.

Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.