ANSYS Autodyn User's 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 2025 R1
January 2025

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

Copyright and Trademark Information

© 2025 ANSYS, Inc. Unauthorized use, distribution or duplication is 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. Application Overview 15
1.1. Autodyn Application Layout 16
1.2. Autodyn Toolbar 16
1.3. Navigation Bar 18
1.4. Dialog Panel and Dialog Windows 19
1.5. Command Line Panel 22
1.6. Undock Command Line 25
1.7. Command File Window 26
1.8. Message Panel 27
1.9. HTML Review Windows 28
1.10. File Names 29
1.11. Initialization Settings 29
1.12. Autodyn in Workbench 34
1.13. Autodyn Solver Options in Workbench 42
2. Pull Down Menus 45
2.1. Pull-Down Menu - File 46
2.1.1. Pull-Down Menu - File - New 48
2.1.2. Pull-Down Menu - File - Manage Folders 50
2.1.2.1. Pull-Down Menu - File - Manage Folders - New 51
2.1.3. Pull-Down Menu - File - Save As 51
2.1.4. Pull-Down Menu - File - Save Project 52
2.1.5. Pull-Down Menu - File - Save As Version 53
2.1.6. Pull-Down Menu - File - Export to Version 54
2.1.7. Pull-Down Menu - File - Export PostScript 55
2.1.8. Pull-Down Menu - File - Export VRML 60
2.2. Pull-Down Menu - Import 62
2.2.1. Pull-Down Menu - Import - True*Grid*® 63
2.2.2. Pull-Down Menu - Import - ICEM 66
2.2.3. Pull-Down Menu - Import - LS-DYNA 67
2.2.4. Pull-Down Menu - Import - Nastran 70
2.2.5. Pull-Down Menu - Import - Convert IJK to Unstructured 72
2.3. Pull-Down Menu - Setup 74
2.3.1. Pull-Down Menu - Setup - Description 76
2.3.2. Pull-Down Menu - Setup - Symmetry 77
2.4. Pull-Down Menu - Execution 78
2.4.1. Pull-Down Menu - Execution - Batch Runner 79
2.5. Pull-Down Menu - View 80
2.5.1. Pull-Down Menu - View - Slideshow - Setup 82
2.5.2. Pull-Down Menu - View - Performance profiler 83
2.6. Pull-Down Menu - Options 84
2.6.1. Pull-Down Menu - Options - Interactivity 85
2.6.2. Pull-Down Menu - Options - Clipping Planes 86
2.6.3. Pull-Down Menu - Options - Default Settings 86
2.6.4. Pull-Down Menu - Options - Pause License 87
2.6.5. Pull-Down Menu - Options - Create User-Subroutine Project 88
2.7. Pull-Down Menu - Help 89
3. Plots 91
3.1. Plots - Select Contour Variable 93
3.2. Plots - View range and slices (Structured) 94

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

3.3. Plots - View range and slices (SPH/Unstructured) 95
4. Settings 97
4.1. Settings - Display 99
4.2. Settings - Grid 105
4.3. Settings - Legend 106
4.4. Settings - Node/Element Numbers 107
4.5. Settings - Materials 108
4.5.1. Settings - Materials - Modify Material Color 110
4.6. Settings - Contour 111
4.6.1. Settings - Contour - Profile Plot 114
4.7. Settings - Velocity vector 116
4.8. Settings - Gauge Point 117
4.9. Settings - Boundary 118
4.10. Settings - Joins 119
4.11. Settings - Axes 120
4.12. Settings - Detonation 121
4.13. Settings - Eroded Nodes 121
4.14. Settings - Unused Regions 122
4.15. Settings - Polygons 123
4.16. Settings - Beam Sections 124
4.17. Settings - Shell Thickness 127
4.18. Settings - Material Direction 128
4.19. Settings - Marker 130
4.20. Settings - Shell Normal 132
4.21. Settings - Part Highlights 133
5. History 135
5.1. History - Single Variable Plot 137
5.2. History - Reduce 138
5.3. History - Write Text Output 139
5.4. History - Multiple Variable Plot 140
5.4.1. History - Multiple Variable Plot - Set All Plots 141
5.4.2. History - Multiple Variable Plot - Modify Selected Plot 142
5.5. History - Min/max 143
5.6. History - Set Scales 144
5.7. History - Set Plot Styles 145
5.8. History - Set Legend Position 146
5.9. History - Load 146
6. Slides 149
6.1. Slides Toolbar 152
6.1.1. Slides Setup 152
6.1.2. Capture Current Image 152
6.1.3. Capture Sequence 153
6.1.4. Text Slide 154
6.2. View Slides 156
7. Materials 161
7.1. Materials - New/Modify 163
7.2. Materials - Copy 167
7.3. Materials - Delete 168
7.4. Materials - Load 169
8. Initial Conditions 171
8.1. Initial Conditions - New/Modify 173

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

8.2. Initial Conditions - Explicit Model Initialization 174
8.3. Initial Conditions - Delete 178
9. Boundaries 179
9.1. Boundaries - New/Modify 180
9.2. Boundaries - Delete 181
9.3. Analytical Blast Boundary Condition 181
9.3.1. User Interface 182
9.3.2. Theory 183
9.3.2.1. Pressure Load Evaluation Without Clearing 183
9.3.2.2. Clearing 'On' 184
9.3.2.3. BLAST.LOAD Variable 185
9.3.2.4. Limitations 185
9.3.2.5. Example – Small Scale Urban Blast 186
9.3.3. References 190
10. Parts 191
10.1. Parts - New 197
10.1.1. Parts - New - Select Predef (2D) 199
10.1.2. Parts - New - Select Predef (3D) 207
10.1.3. Parts - New - Select Predef (3D Shell) 214
10.1.4. Parts - New - Define Zoning (2D) 217
10.1.5. Parts - New - Define Zoning (3D) 226
10.1.6. Parts - New - Define Zoning (3D Shell) 234
10.1.7. Parts - New - Fill Part (2D) 238
10.1.8. Parts - New - Fill Part (3D) 240
10.1.9. Parts - New - Fill Part (3D Shell) 243
10.2. Parts - Copy 244
10.3. Parts - Delete 245
10.4. Parts - IJK Range 245
10.5. Parts - Activation 246
10.6. Parts - Rename 246
10.7. Parts - Print 247
10.8. Parts - Pack (Fill) 247
10.9. Parts - Zoning 248
10.9.1. Parts - Zoning - Box 252
10.9.2. Parts - Zoning - Cylinder 254
10.9.3. Parts - Zoning - Sphere 256
10.9.4. Parts - Zoning - Ogive 257
10.9.5. Parts - Zoning - Quad 259
10.9.6. Parts - Zoning - Hex 260
10.9.7. Parts - Zoning - Frag/Bric (Page 1) 261
10.9.8. Parts - Zoning - Frag/Bric (Page 2) 263
10.9.9. Parts - Zoning - Transition 265
10.9.10. Parts - Zoning - Node 266
10.9.11. Parts - Zoning - Line 267
10.9.12. Parts - Zoning - Surface 268
10.9.13. Parts - Zoning - Volume 270
10.9.14. Parts - Zoning - Translate 271
10.9.15. Parts - Zoning - Rotate 272
10.9.16. Parts - Zoning - Scale 273
10.9.17. Parts - Zoning - Copy 274
10.9.18. Parts - Zoning - Delete 275

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

10.9.19. Parts - Zoning - Save 276
10.9.20. Parts - Zoning - Restore 277
10.9.21. Parts - Zoning - Refine 277
10.9.22. Parts - Zoning - Fill Block 278
10.9.22.1. Parts - Zoning - Fill Block Implicit 279
10.9.22.2. Parts - Zoning - Fill Block Explicit 280
10.10. Parts - Fill 282
10.10.1. Parts - Fill - Block 284
10.10.2. Parts - Fill - I / J / K Plane 286
10.10.3. Parts - Fill - Node 287
10.10.4. Parts - Fill - Unused 288
10.10.5. Parts - Fill - Composite 289
10.10.5.1. Parts - Fill - Composite - New 290
10.10.5.2. Parts - Fill - Composite - Modify 291
10.10.6. Parts – Fill – Beam 292
10.10.6.1. Available Beam Cross Sections 292
10.10.6.2. Spring Cross Section 295
10.10.6.3. Damper cross section 295
10.10.7. Parts - Fill - Multi-Fill 296
10.10.8. Parts - Fill - Datafile 298
10.10.9. Parts - Fill - Part Fill 302
10.10.10. Parts - Fill - Block Remap 303
10.11. Parts - Boundary 307
10.11.1. Parts - Boundary - Block 309
10.11.2. Parts - Boundary - I / J / K Plane 310
10.11.3. Parts - Boundary - Node 311
10.11.4. Parts - Boundary - Clear 312
10.11.5. Parts - Boundary - Apply Interactively 312
10.11.6. Parts - Bound - Clear Interactively 313
10.12. Parts - Gauges 314
10.12.1. Parts - Gauges - Add 316
10.12.2. Parts - Gauge - Move 317
10.12.3. Parts - Gauge - Delete 318
10.12.4. Parts - Gauges - Add Unstructured 319
10.12.5. Parts - Gauge - Move Unstructured 320
10.12.6. Parts - Gauge - Delete Unstructured 321
10.13. Parts - Solvers (Structured) 321
10.13.1. Parts - Solvers (Structured) - Type 322
10.13.2. Parts - Solvers (Structured) - Solution (2D Shells) 323
10.13.3. Parts - Solvers (Structured) - Solution (3D Shells) 324
10.13.4. Parts - Solvers (Structured) - Solution (Beams) 324
10.13.5. Parts - Solvers (Structured) - Solution (3D Euler-FCT) 325
10.13.6. Parts - Solvers (Structured) - Solution (2D SPH) 326
10.13.7. Parts - Solvers (Structured) - Solution (3D SPH) 327
10.13.8. Parts - Solvers (Structured) - Dezone (2D & 3D Euler) 328
10.13.9. Parts - Solvers (Structured) - Dezone (2D & 3D Lagrange/ALE) 329
10.13.10. Parts - Solvers (Structured) - Jetting 330
10.14. Parts - Solvers (Unstructured) 331
10.14.1. Parts - Solvers (Unstructured) - Solid 331
10.14.2. Parts - Solvers (Unstructured) - Shell 336
10.14.3. Parts - Solvers (Unstructured) - Beam 337

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

10.15. Parts - Motions 338
10.15.1. Parts - Motions - Block 339
10.15.2. Parts - Motions - I / J / K Plane 340
10.15.3. Parts - Motions - Node 341
10.15.4. Parts - Motions - Modify 341
10.16. Parts - Geometry 342
10.16.1. Parts - Geometry - SPH 343
10.16.2. Parts - Geometry - New/Modify 344
10.16.3. Parts - Geometry - Copy 348
10.16.4. Parts - Geometry - Import Part 348
10.16.5. Parts - Geometry - Translate 350
10.16.6. Parts - Geometry - Rotate 350
11. Components 353
11.1. Components - New 354
11.2. Components - Modify 356
11.3. Components - Delete 357
11.4. Components - Material 357
11.5. Components - Velocity 358
11.6. Components - Initial Conditions 359
11.7. Components - Apply Boundary 360
11.8. Components - Clear Boundary 360
11.9. Components - Translate 361
11.10. Components - Rotate 362
11.11. Components - Scale 363
11.12. Components - Copy 363
12. Groups 365
12.1. Groups - New 374
12.2. Groups - Rename 375
12.3. Groups - Copy 375
12.4. Groups - Delete 376
12.5. Groups - Apply 377
12.6. Groups - Clear 377
12.7. Groups - Material 378
12.8. Groups - Velocity 379
12.9. Groups - Initial Condition 380
12.10. Groups - Translate 381
12.11. Groups - Rotate 382
12.12. Groups - Scale 383
13. Joins 385
13.1. Joins - Join 389
13.2. Joins - Unjoin 390
13.3. Joins - Matrix 391
13.4. Joins - Merge Joined Nodes 392
13.5. Joins - Bonded Face Connections 392
13.5.1. Verification of Bond Status 395
13.5.1.1. Using the `Prt` File Output 395
13.5.1.2. Using Preprocessing Output 396
13.5.1.3. Using Post Processing Output 398
13.5.1.4. Bonded Contact Warning Messages 398
14. Interactions 401
14.1. Interactions - Lagrange/Lagrange 402

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

14.1.1. Interactions - Lagrange/Lagrange - Introduction 402
14.1.2. Interactions - Lagrange/Lagrange - Create 405
14.1.3. Interactions - Lagrange/Lagrange - Add 406
14.1.4. Interactions - Lagrange/Lagrange - Remove 407
14.1.5. Interactions - Lagrange/Lagrange - Matrix 408
14.1.6. Interactions - Lagrange/Lagrange - Range 409
14.1.7. Interactions - Lagrange/Lagrange - Friction 410
14.1.8. Interactions - 2D Gap Contact Options 412
14.1.9. Interactions - Lagrange/Lagrange - 2D Trajectory Contact Options 415
14.1.10. Interactions - Lagrange/Lagrange - 3D Gap Contact Options 418
14.1.11. Interactions - Lagrange/Lagrange - 3D Trajectory Contact Options 423
14.2. Interactions - Euler/Lagrange (2D) 431
14.2.1. Interactions - Euler/Lagrange - Introduction 435
14.2.2. Interactions - Euler/Lagrange (2D) - New Polygon 437
14.2.3. Interactions - Euler/Lagrange (2D) - Velocity 438
14.2.4. Interactions - Euler/Lagrange (2D) - Porosity 438
14.2.5. Interactions - Euler/Lagrange (2D) - New Interaction 439
14.3. Interactions - Euler/Lagrange (3D) 440
14.3.1. Interactions - Euler/Lagrange (3D) - Type 442
14.3.2. Interactions - Euler/Lagrange (3D) - Select 446
14.3.3. Interactions - Euler/Lagrange (3D) - Thickness 447
14.3.4. Interactions - Euler/Lagrange (3D) - Joins 448
14.3.5. Interactions - Euler/Lagrange (3D) - Weak Coupling 449
15. Detonation (2D) 453
15.1. Detonation (2D) - Point 455
15.2. Detonation (2D) - Line 456
15.3. Detonation (2D) - Circle 458
15.4. Detonation (2D) - Manual 459
16. Detonation (3D) 461
16.1. Detonation (3D) - Point 463
16.2. Detonation (3D) - Plane 464
16.3. Detonation (3D) - Cylinder 466
16.4. Detonation (3D) - Sphere 467
16.5. Detonation (3D) - Manual 469
17. Parallel Processing 471
17.1. Parallel - Sub-domains 473
17.2. Parallel - Task Assignments 474
17.3. Parallel - Automatic Decomposition 477
17.4. Parallel - Automatic Decomposition of Euler parts 478
17.5. Parallel - Decomposition of beam parts 482
17.6. Parallel – Fully Automatic Decomposition 483
18. Output 485
18.1. Output - Part Summary Variables 491
18.2. Output - Results Files 493
19. User Variables 497
19.1. User Variables - Add/Modify 498
20. Controls 499
20.1. Controls - Mass Scaling 507
20.2. Controls - Global Erosion 510
21. Usage in Ansys 511
21.1. Autodyn Walkthrough Using Explicit Dynamics 512

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

21.2. Autodyn Walkthrough Using Meshing 519
22. Additional Information 525
22.1. Running Autodyn on Windows Platforms 525
22.2. Running Autodyn in Batch Mode on Linux Platforms 527
22.2.1. Running Autodyn using the AUTODYNWRAPPER 527
22.2.2. Running Autodyn using the **autodyn** Script 529
22.3. Files Created by Autodyn 529
22.4. Explicit Dynamics Material Models 530
22.5. Explicit Dynamics Theory 530
22.6. Autodyn Material Models 531
22.6.1. Equation of State 531
22.6.1.1. Tillotson EOS 531
22.6.1.1.1. Regions of Interest 532
22.6.1.1.2. Regional Formulations 533
22.6.1.2. Puff EOS 534
22.6.1.2.1. Regions of Interest 534
22.6.1.2.2. Regional Formulations 535
22.6.1.3. Two-Phase EOS 536
22.6.1.3.1. Single Phase Region 537
22.6.1.3.2. Two-Phase Region 537
22.6.1.4. Sesame EOS (Tables) 538
22.6.1.5. Explosive EOS 539
22.6.1.5.1. Lee-Tarver Model 539
22.6.1.5.2. Powder Burn EOS 541
22.6.1.6. Rigid 544
22.6.2. Material Strength 544
22.6.2.1. Von Mises Model 545
22.6.2.2. Piecewise JC Model 546
22.6.2.3. Beam Resistance Model 546
22.6.3. Failure Models 547
22.6.3.1. Material Stress Failure 548
22.6.3.2. Material Strain Failure 549
22.6.3.3. Material Stress/Strain Failure 550
22.6.3.4. Cumulative Damage Model 550
22.6.3.5. Orthotropic Softening 552
22.6.3.6. Tsai - Hoffman - Hill Model 553
22.6.3.7. MO Granular Failure Model 554
22.6.4. Material Cutoffs 556
22.7. References 557
Index 559

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information
of ANSYS, Inc. and its subsidiaries and affiliates.

List of Figures

6.1. Components of the Viewer's user interface 156
9.1. Overpressure-Time Curve 184
9.2. Effect of the Clearing Option on Pressure Load 185
9.3. Small Scale Blast Experiment 186
9.4. Side-On Overpressure 188
9.5. Reflected Overpressure 189
9.6. Channelled Street Blast Results 189
12.1. Example use of Delete Elements Group Operation 371
12.2. Example use of Split nodes to create conformal sliding interface 372
12.3. Example use of Merge nodes to remove joins. 373

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information
of ANSYS, Inc. and its subsidiaries and affiliates.

List of Tables

9.1. Peak incident overpressure and positive phase impulses at the gauge on the front of the building for a spherical explosion without the concrete blocks 187
9.2. Peak reflected overpressure and positive phase impulses at the gauge on the front of the building without the concrete blocks 187
9.3. Peak overpressures and positive phase impulses at both the front and side gauges when the concrete blocks are present 188
10.1. Material Modeling Options Available with NBS-Tet Elements 334

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information
of ANSYS, Inc. and its subsidiaries and affiliates.

Chapter 1: Autodyn Application Overview

Ansys Autodyn is an explicit analysis tool for modeling nonlinear dynamics of solids, fluids, gas, and their interaction. With a fully integrated, easy-to-use graphical interface enabling the set up, running, and postprocessing of problems, Autodyn offers:

- Finite element solvers for computational structural dynamics (FE).
- Finite volume solvers for fast transient Computational Fluid Dynamics (CFD).
- Mesh-free particle solvers for high velocities, large deformation, and fragmentation (SPH).
- Multi-solver coupling for multi-physics solutions including coupling between FE, CFD, and SPH.
- A wide suite of material models incorporating constitutive response and coupled thermodynamics.
- Serial and parallel computation on shared and distributed memory systems.

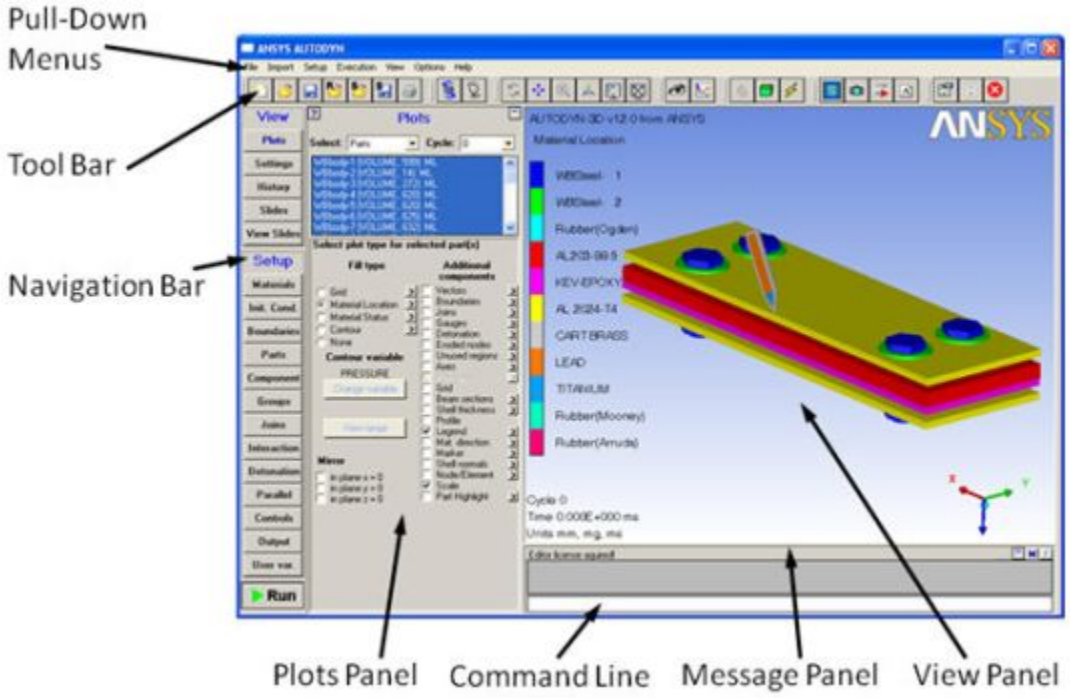
Autodyn has been used in a vast array of projects and nonlinear phenomena. Here is a sample of real projects where it has been used:

- Optimization and design of armor and anti-armor systems.
- Designing mine protection schemes for personnel carriers.
- Building protection measures and insurance risk assessment for blast effects in city centers.
- Aircraft impact risk assessment for power stations.
- Performance studies of oil-well perforating charges.
- Decommissioning of offshore platforms.
- Designing the shielding system on the International Space Station.
- Safety assessment of particle accelerators.
- Characterization of materials subjected to high dynamic loading.
- Drop test of electronics devices.
- Nuclear reactor safety.
- Nuclear waste transportation safety.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

1.1. Autodyn Application Layout

Autodyn provides a user-friendly graphical user interface combining pre-processor, solver, and post-processor in the same windows environment.



A single executable performs simulations in both two and three dimensions. The user interface is laid out with buttons running horizontally along the top and vertically down the left hand side of the main window. The horizontal group of buttons is referred to as the toolbar and the vertical group as the navigation bar. The toolbar and navigation bar offer quick access to options also available through the pull-down menus.

The main window is composed of a number of panels (View, Dialog, Message, and Command Line).

Caution:

Scaling is not recommended when using the Autodyn user interface. Using the scaling option will result in distorted menus and panels that may prevent you from seeing the entire contents.

1.2. Autodyn Toolbar

The toolbar (and the navigation bar) provide shortcuts to options available in the pull-down menus.

Button	Description
	Create a new model (this functionality is available only when Autodyn is started up outside Ansys Workbench)

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Button	Description
	Open an existing model (this functionality is available only when Autodyn is started up outside Ansys Workbench)
	Save current model to current filename (this functionality is available only when Autodyn is started up outside Ansys Workbench)
	Open a results file
	Open a settings file
	Save current plot view parameters to a settings file
	Print
	Transform object default to on
	Transform light source
	Rotate model
	Translate model
	Zoom function
	Set view brings up new window so that exact view can be set
	Reset model
	Fit model to view panel
	Examine model
	Profile window
	Toggle wireframe mode (on/off)
	Perspective on/off
	Hardware acceleration on/off

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Button	Description
	Setup slide show
	Create image of current view
	Record slideshow
	Create textslide
	Show/hide Navigation Bar
	Manual/Automatic refresh
	Refresh screen
	Stop all plotting

1.3. Navigation Bar

The Navigation Bar contains two groups of buttons.

The top group of buttons (View) let you control what you see in the View Panel. From here you can access and modify plot settings, view history plots, and create or view slide shows / animations :

The lower group of buttons (Setup) let you access dialog panels to set up your model. Starting at the top button (Materials) and working down to the Run button, you can build your model in the most efficient and logical way :

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

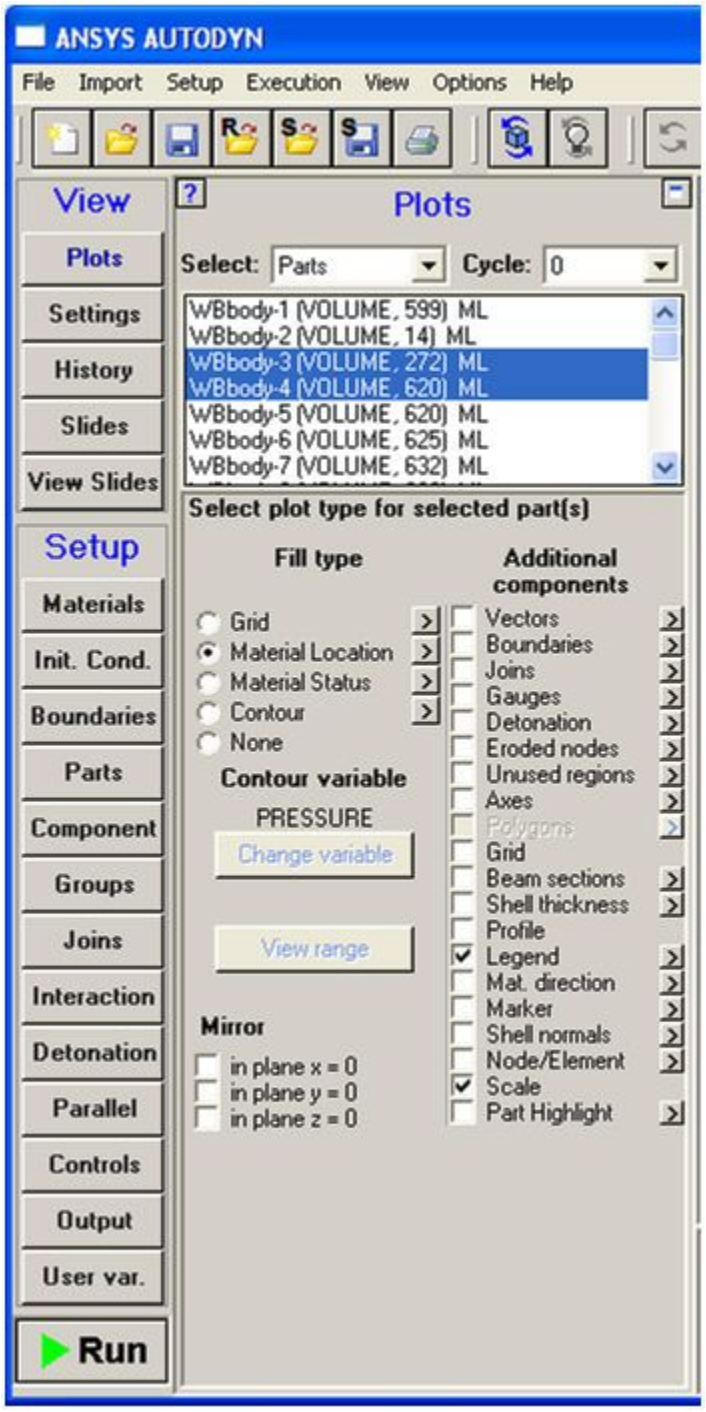
The Problem Setup section explains this procedure in detail:

The Run button at the foot of the bar is used to run (execute) problems :

1.4. Dialog Panel and Dialog Windows

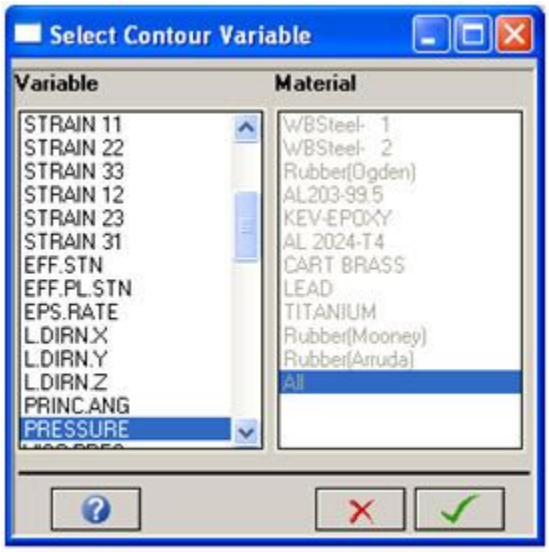
When you select an option from the Navigation Bar, dialog for that option appears in the Dialog panel :

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



The Dialog panel generally contains input fields and buttons that access further input options. Clicking a button in the dialog panel will display further dialog either within the dialog panel or in a new Dialog Window that is opened :

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

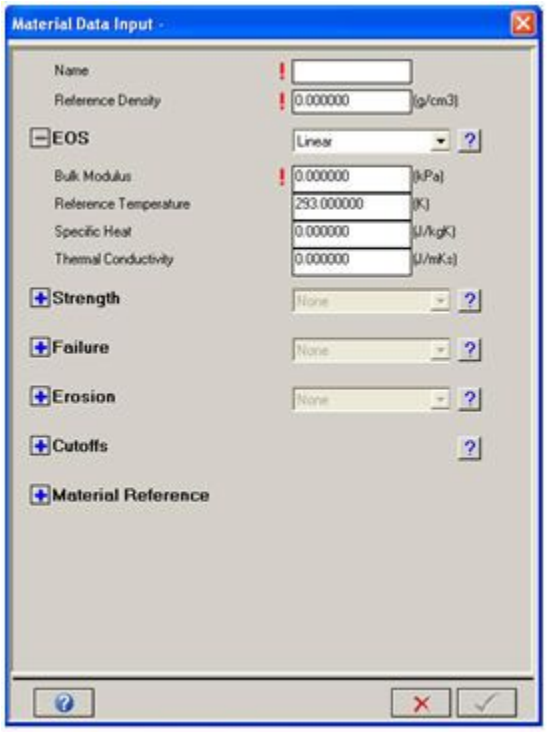


At the bottom of all Dialog Windows there are three buttons.

The button with a question mark provides help on the current windows function.

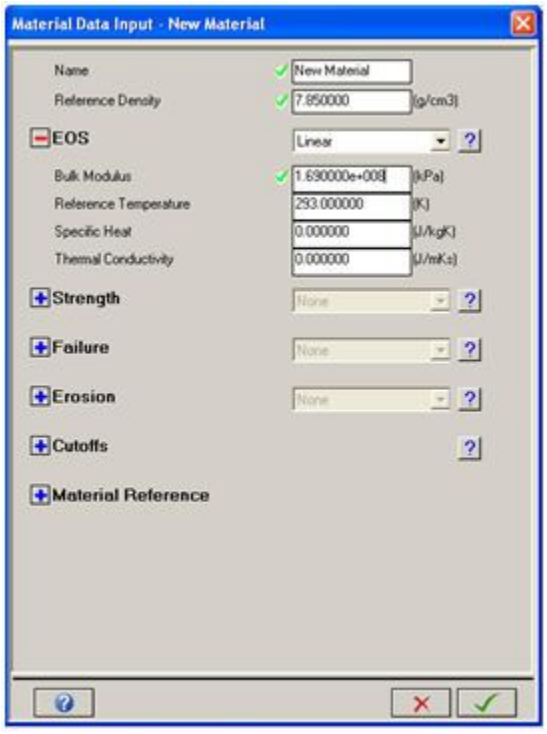
The two other buttons are (Cancel) and (OK). Clicking the Cancel button will close the window, without applying any of the changes that you made within it. The OK button closes the window and applies your changes.

In certain cases, an Apply button will be present. This provides the ability to apply any changes made within the window without closing the window.



In the dialog panel or dialog windows, input data that is required for a task to complete is marked (above). When you enter an appropriate value, the mark changes to (below). The OK button will remain inactive until you provide appropriate values for all required fields:

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



If a drop down menu contains a large number of elements, only the first 30 are shown. Use the cursor keys to scroll down to and select items outside of this range.

1.5. Command Line Panel

The command language provides the capability to:

- Record operations made in the user interface
- Modify/Playback/repeat operations
- Load operations previously saved to file
- Set up models using the command language.

This feature has Beta status and you must activate it by adding the following lines above the Additional User Title option in the `autodyn.ini` file.

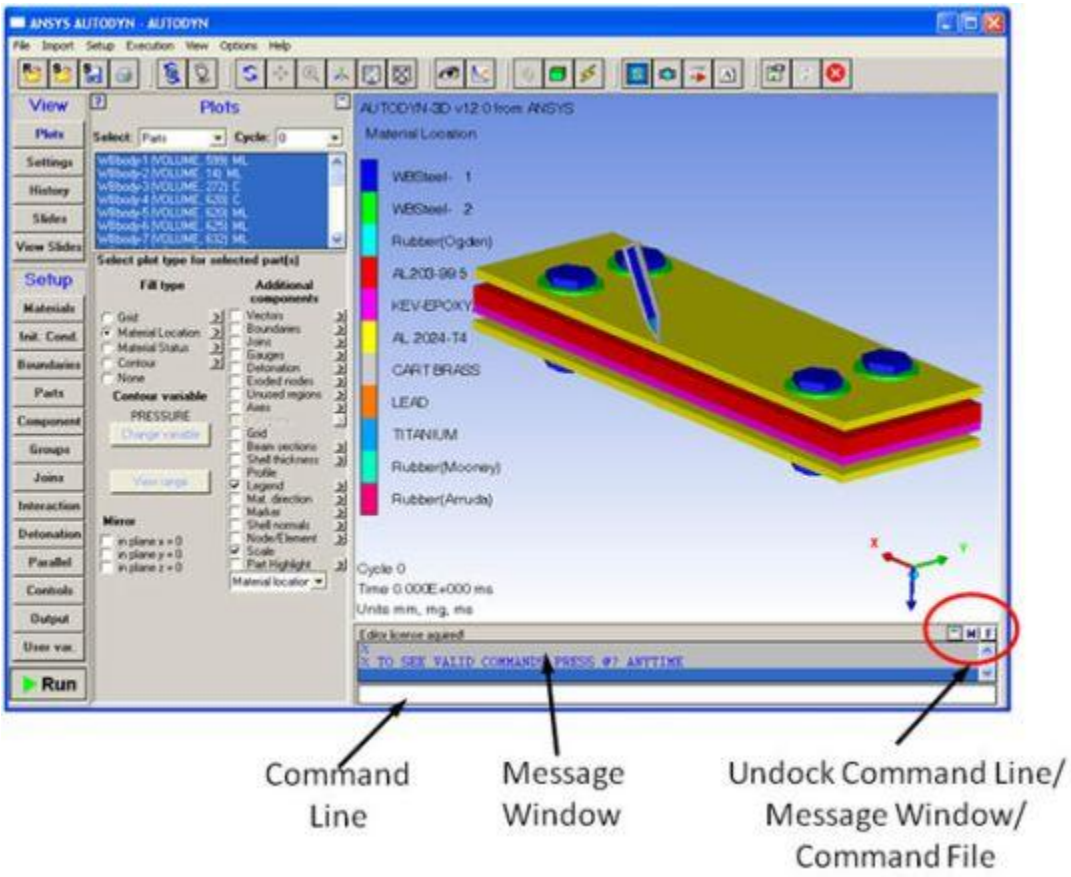
```
command_line_on
1
```

Caution:

By activating this flag you accept that this is a Beta option and therefore may not be as stable or robust as expected in a general release version.

The command line is initially inactive until a model is either loaded or created. To the right of the message line there are two buttons (in addition to the Message window button) that are related to the command line.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Commands can be typed directly into the command line with multiple commands being concatenated by using a ';' between the commands, i.e. PARTS:;TARGET to select a part named TARGET. Useful/help information is output to the Message/Output window. Note however that the command language is more easily used when the command window is undocked from the main user interface.

The command line creates two command files:

autodyn.adc

This file is located in the directory `C:\Documents and Settings\username\Local Settings\Temp` and is overwritten every time a new Autodyn session is started.

ident.adc

This file is created every time a new model is created. If the model is subsequently modified in a different Autodyn session the new commands are appended to this file. Therefore the ident.adc should contain all commands that have been executed on the model since it was first created.

A typical series of commands will look similar to this:

```
PARTS:
target
  @ZONING
  #OPERATION TYPE = Zoning
    #ZONING?= Predef
      #PREDEF GEOMETRY?= Box
        X Origin?=?0.0000000E+00
        Y Origin?=?0.0000000E+00
        Z Origin?=?0.0000000E+00
```

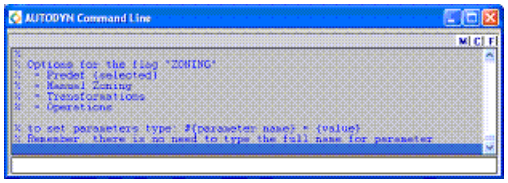
Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

```
        DX?= 10.00000
        DY?= 10.00000
        DZ?= 10.00000
        #GRADING I?= NO
        #GRADING J?= NO
        #GRADING K?= NO
  @DO
  @FILL
  #OPERATION TYPE = Fill
    #FILL BY INDEX = Block
      I min?=?1
      I max?= 11
      J min?=?1
      J max?= 11
      K min?=?1
      K max?= 11
      #FILL WITH INITIAL CONDITION SET? = No
      #FILL MATERIAL? = AL 2024
      Density = 2.785000
      Int. Energy = 0.0000000E+00
      X Velocity? = 0.0000000E+00
      Y Velocity? = 0.0000000E+00
      Z Velocity? = 0.0000000E+00
      #RADIAL VELOCITY = No
      #ANGULAR VELOCITY? = No
@DO
```

The 'PARTS:' command word is the name of the current command structure. (When other parts of the interface are covered by the command line, each of the buttons on the Navigation panel will be a command structure (that is, Materials, Boundaries, and so on).)

The name of the part currently selected is then given, in this case it is called 'target'. In this example the command being executed is a zoning command followed by a fill command- @ indicates a command. To see the current list of available commands '@?' can be typed at any time.

A # denotes an option. In the above example, #zoning enables you to choose from the list of zoning options. Typing '?#' will give you the list of options for zoning the options are shown below:



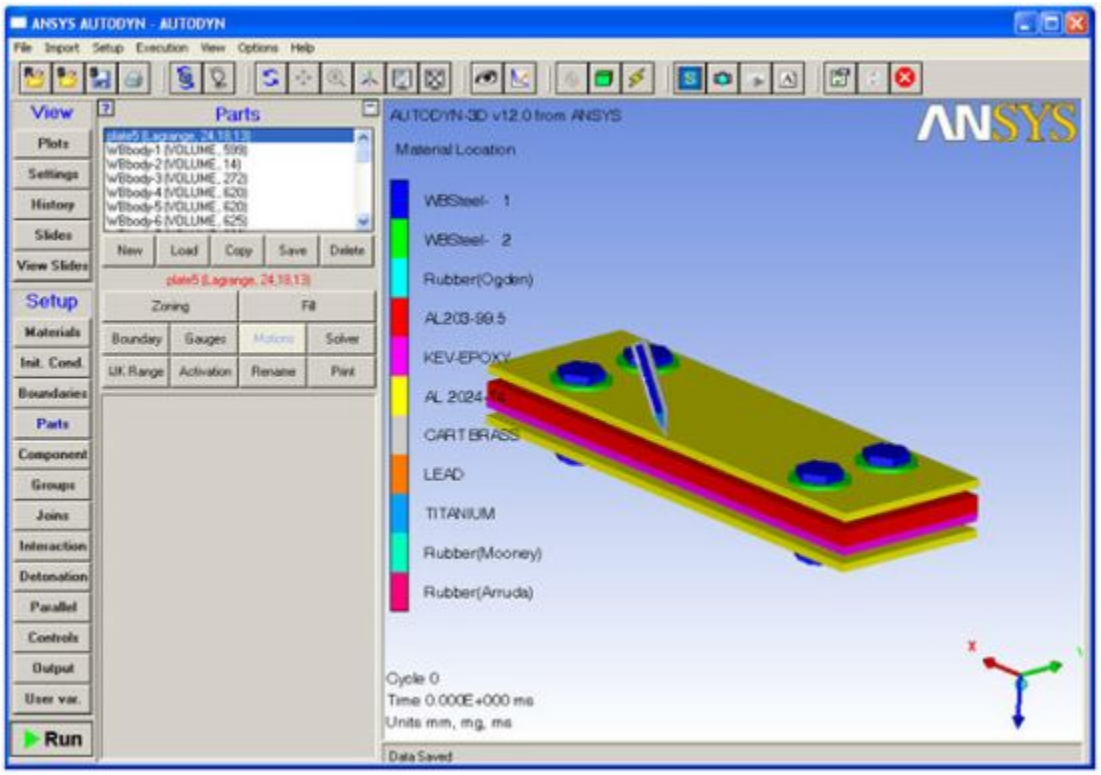
For the example given here, predef is chosen as the option. The predef zoning option has a sub-option '#PREDEF GEOMETRY' that enables you to choose the predef you want to zone. In this simple example, the box predef is chosen and the required parameters have been set.

When all the parameters have been set the '@DO' command is issued, which executes the command.

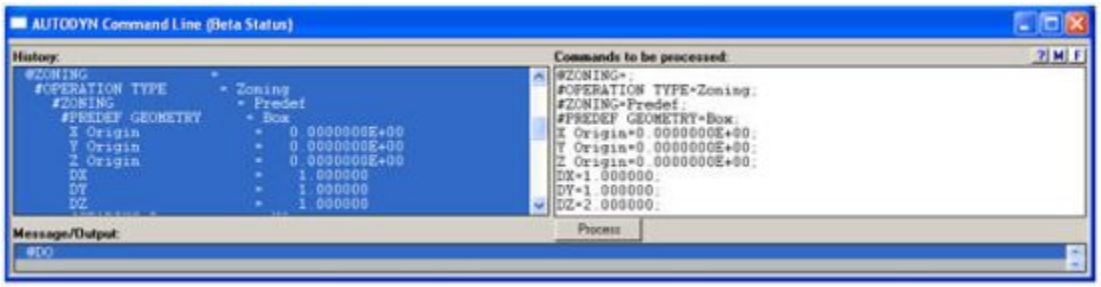
The easiest way to use the command line and become familiar with the command structure is to use the user interface to zone and fill parts as usual. The commands will then be recorded and can then be edited and played back as described above. Also, as can be seen in the picture above some guidance is given in the history window for the syntax used to set options and parameters etc. should you choose to type in the commands directly.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

1.6. Undock Command Line



Click this button to detach the command line panel from the main user interface and open it in a separate dialog window. If this window is then closed, the Command Line docks back into the main user interface.



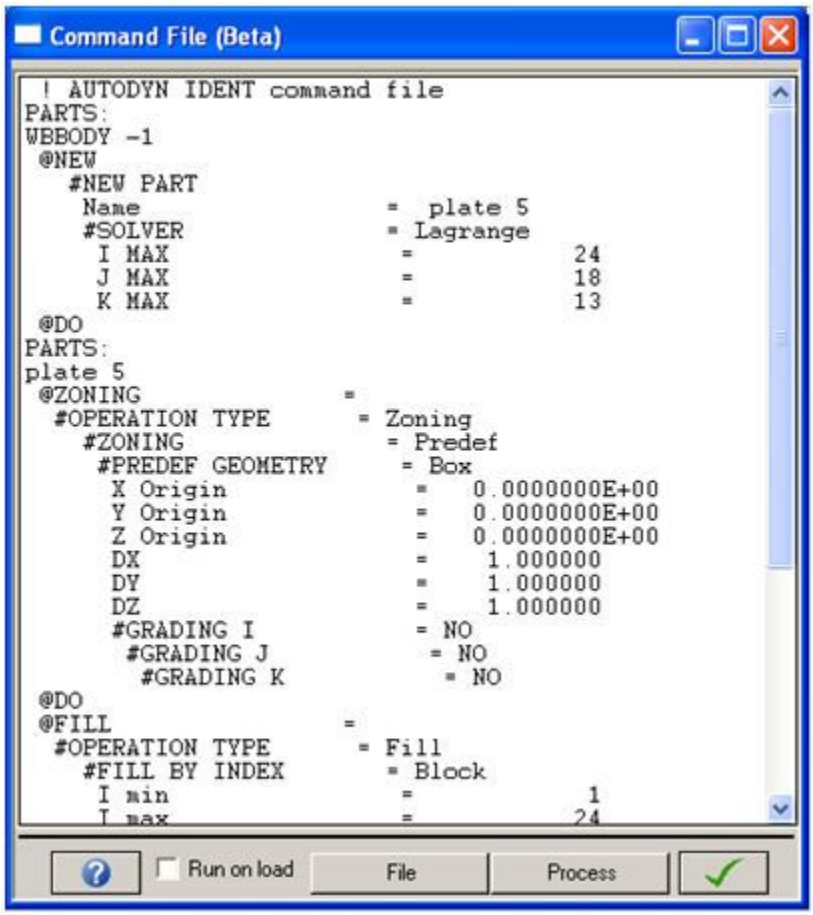
When the command line is detached from the main user interface the command dialog panel contains three separate lists. **History** shows the command history on the left hand side and contains all valid commands executed in the current session. Multiple commands can be selected from this list to be replayed. The **Message/Output** panel is shown at the bottom. This provides you with help if you request it and gives any error messages when invalid commands are performed.

The **Commands to be processed** list behaves the same as the command line when docked into the main user interface but is much more user friendly and enables multiple lines to be input before clicking the process button. Any commands selected from the History list are copied into the **Commands to be processed** panel and can then be edited before being processed.

The example above shows how you can replay a zoning operation. In this case the DZ parameter is modified before the command is re-processed.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

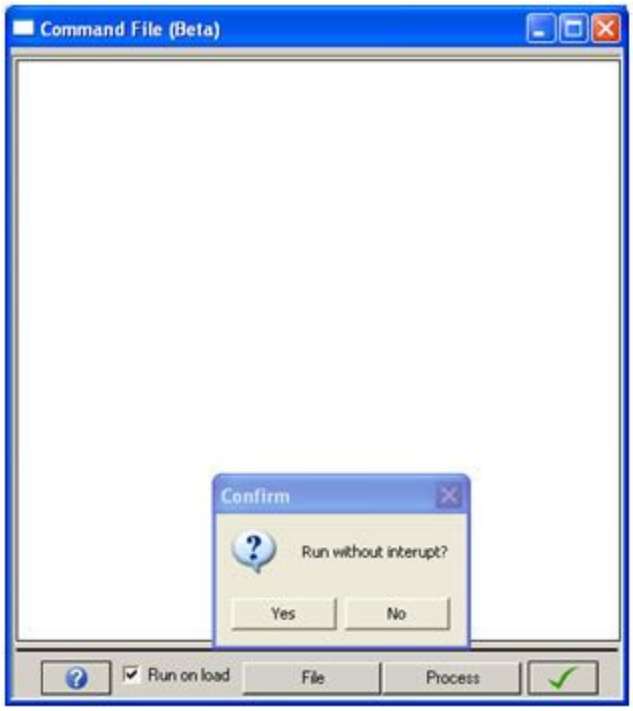
1.7. Command File Window



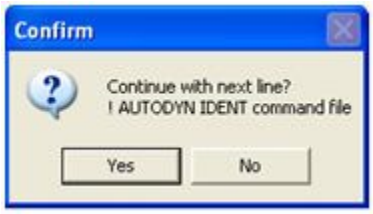
The command file window enables you to open an existing Autodyn command file using the **File** button.

The commands can be executed/modified in a number of ways. Select all or a number of the commands and click the **Process** button. Alternatively, if you want to modify the commands before execution, simply select the required commands. The commands are then copied to either the single command line or the **Commands to be processed** list (if the command line is not docked to the main user interface). The commands can then be modified before execution by either pressing return on the single command line or the process button in the Command line panel.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



If the **Run on Load** option is selected the commands in the selected file can automatically be run on loading the file, or alternatively you are given the option to interactively step through the commands.

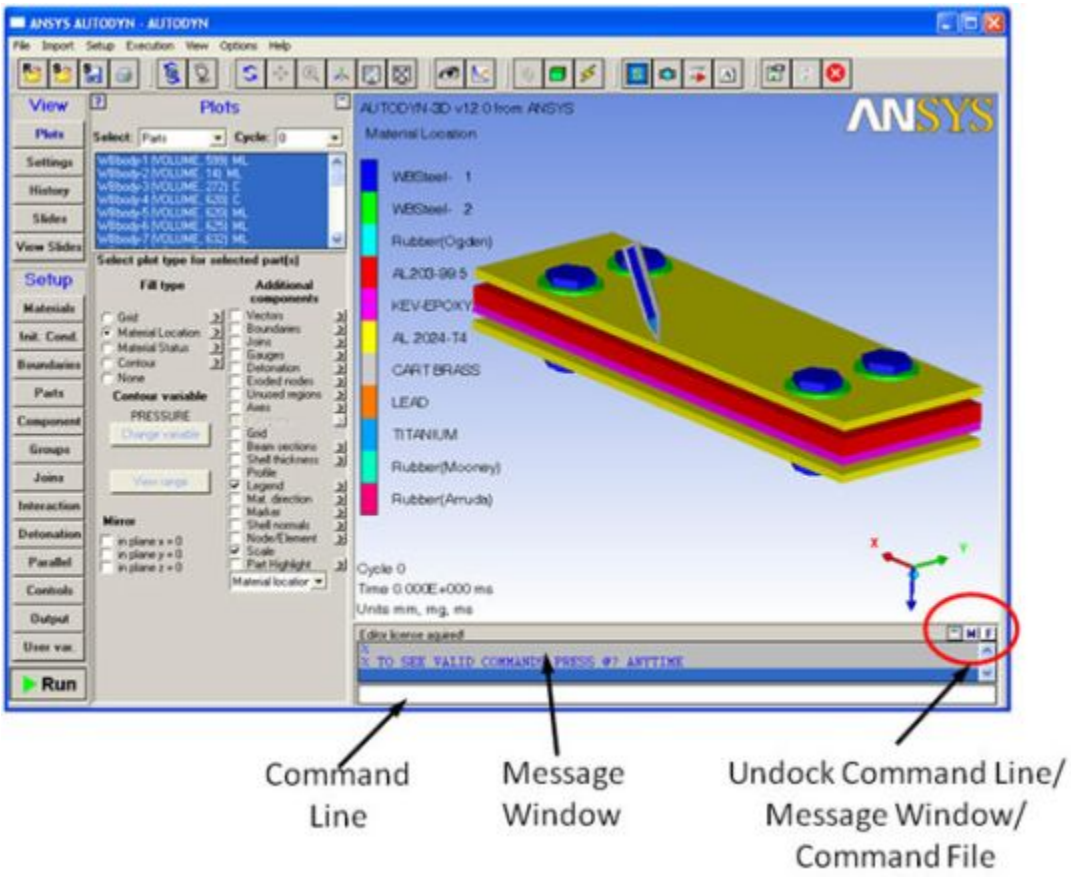


If stepping through the command file, you are shown the next command before processing it. If at any stage **No** is selected, Autodyn stops reading the file.

1.8. Message Panel

The message panel shows a variety of messages during problem set up, execution, and viewing.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

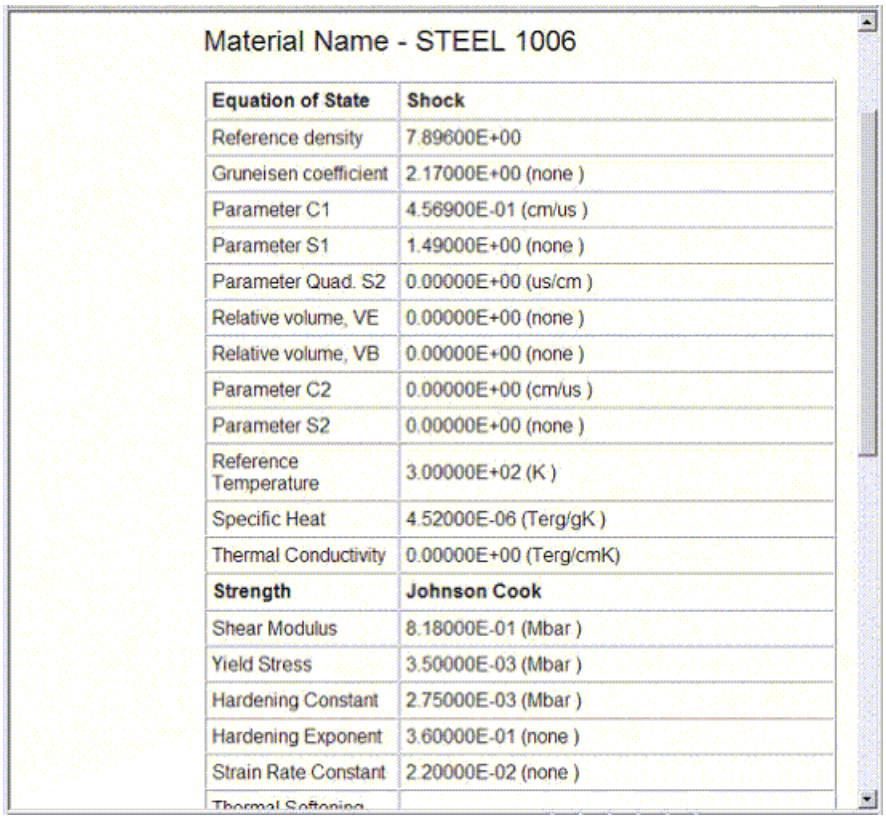


Clicking the **M** box at the top right corner of the message panel will open a scrollable window showing all the messages displayed during your session.

1.9. HTML Review Windows

While working on a model, you may want a summary of particular data such as material properties, detonation point or gauge point locations. These are created as an HTML document and displayed in a browser window. You can easily insert these documents into reports and publications.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Material Name - STEEL 1006

Equation of State	Shock
Reference density	7.89600E+00
Gruneisen coefficient	2.17000E+00 (none)
Parameter C1	4.56900E-01 (cm/us)
Parameter S1	1.49000E+00 (none)
Parameter Quad. S2	0.00000E+00 (us/cm)
Relative volume, VE	0.00000E+00 (none)
Relative volume, VB	0.00000E+00 (none)
Parameter C2	0.00000E+00 (cm/us)
Parameter S2	0.00000E+00 (none)
Reference Temperature	3.00000E+02 (K)
Specific Heat	4.52000E-06 (Terg/gK)
Thermal Conductivity	0.00000E+00 (Terg/cmK)
Strength	**Johnson Cook**
Shear Modulus	8.18000E-01 (Mbar)
Yield Stress	3.50000E-03 (Mbar)
Hardening Constant	2.75000E-03 (Mbar)
Hardening Exponent	3.60000E-01 (none)
Strain Rate Constant	2.20000E-02 (none)
Thermal Softening	

Note:

If a browser does not launch to display a review window, ensure that files of type `htm` are associated with a browser, and that no errors occur when launching the browser from the operating system.

1.10. File Names

Autodyn file names have the underlying format of *`Name_information.extension`*.

Examples:

- A cycle zero Save file would be named `myfilename_0.ad`
- The Save file for cycle 12500 would be named `myfilename_12500.ad`.

1.11. Initialization Settings

Autodyn is installed with a pre-configured file called `autodyn.ini`, which is in your Application Data directory, in the subdirectory `Ansys\v251\AUTODYN`. These are the contents of the file after installation:

```
AUTODYN_initialization_file
application_window_width
1000
application_window_height
700
message_window_visibility
```

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

```
0
manual_refresh_default
0
background_color
1
hardware_acceleration
1
reset_plot_flags
0
plot_wireframe
0
plot_perspective
0
model_interactivity_default
0
graded_background
1
OpenGL_for_slides
1
remove_joined_faces
0
open_new_browser_window
1
command_line_on
1
enable_beep
0
mouse_option_rotate
2
keyboard_option_rotate
0
mouse_option_zoom
3
keyboard_option_zoom
0
mouse_option_translate
2
keyboard_option_translate
2
mouse_option_toolbar
1
keyboard_option_toolbar
0
mouse_option_examine
1
keyboard_option_examine
3
mouse_option_boxzoom
3
keyboard_option_boxzoom
2
use_prepost_only
0
additional_user_title
```

You can alter the contents of this file to customize the startup of Autodyn.

You can use any text editor to make your changes (for example, Notepad or MS Word).

You can set the following parameters in this file.

Application window width

Sets the width of the Autodyn application window at startup.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Application window height

Sets the height of the Autodyn application window at startup.

Message window visibility

Sets the visibility of the message window on startup.

0

Message window hidden

1

Message window visible

Manual refresh default

Sets manual refresh on or off. For large models you may want to have full control on the refreshing of the plot window.

0

Manual refresh off

1

Manual refresh on

Background color

Sets the color of the background for the View Panel on startup.

0

Black background

1

White background

Plot wireframe

Sets the wireframe plotting option on or off on startup.

0

Plot wireframe off

1

Plot wireframe on

Plot perspective

Sets the perspective plotting option on or off on startup.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

0

Plot perspective off

1

Plot perspective on

Hardware acceleration

Sets hardware graphics acceleration on or off on startup.

0

Hardware graphics acceleration off

1

Hardware graphics acceleration on

Model Interactivity default

Sets the default model interaction setting for the toolbar mouse/keyboard combination on startup.

1

Translate model

2

Rotate model

3

Zoom in/out

Reset plot flags

Sets reset plot flags on or off.

- If this flag is on, when you load a new model (or a different cycle of the current model) the plot settings for all Parts will be reset to default values (for example, default view, material locations, and so on).
- If this flag is off, when you load a new model (or a different cycle of the current model) with a model already present, the plot settings (grid, material location, material status, contour) for the current model are migrated to the new model for each Part with the same name.

0

Reset plot flags off

1

Reset plot flags on

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Graded background

Sets graded background on or off. If this switch is on a graded background color is used. If this switch is off a solid black or white color is used depending on Background color setting.

0

Graded background off

1

Graded background on

OpenGL for Slide creation

Sets OpenGL for Slide creation on or off. Using OpenGL during the creation of slides can cause problems for some graphics cards. If you experience any problems creating slides with this switch turned on, try turning it off.

0

OpenGL off for slide creation

1

OpenGL on for slide creation

Browser Options

By default, when Autodyn opens an HTML document (for the summary reviews), the page will be displayed in any browser that is currently open. By setting this flag to 1, a browser will be opened to display the information. This option may not work for all browsers.

Command Line Options

Set this flag to 1 to enable the command line.

Warning Sound Options

Enables or disables the warning beep

Mouse and Keyboard options

These options let you set the mouse and keyboard combinations for interaction. Each action type has a keyboard and an associated mouse option. A default will be used if any are not defined.

Mouse action

0	Not used
1	Left Mouse Button
2	Middle Mouse Button

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

3	Right Mouse Button

Keyboard action

0	None
1	Shift
2	Ctrl
3	Alt

Keyboard action

Mouse Option Rotate + Keyboard Option Rotate	Rotate mouse and keyboard combination
Mouse Option Zoom + Keyboard Option Zoom	Zoom mouse and keyboard combination
Mouse Option Translate + Keyboard Option Translate	Translate mouse and keyboard combination
Mouse Option Toolbar + Keyboard Option Toolbar This is the default combination for the interaction type selected on the toolbar	Toolbar mouse and keyboard combination
Mouse Option Examine + Keyboard Option Examine	Examine mouse and keyboard combination
Mouse Option Box zoom and Keyboard Option Box zoom	Box zoom mouse and keyboard combination

PrePost license usage

By default this switch is set to zero and Autodyn will start up using a solver license. When the switch is set to zero Autodyn will start up using a PrePost license only.

Additional user title

Enables you to add additional information below the top left hand corner of the View Panel.

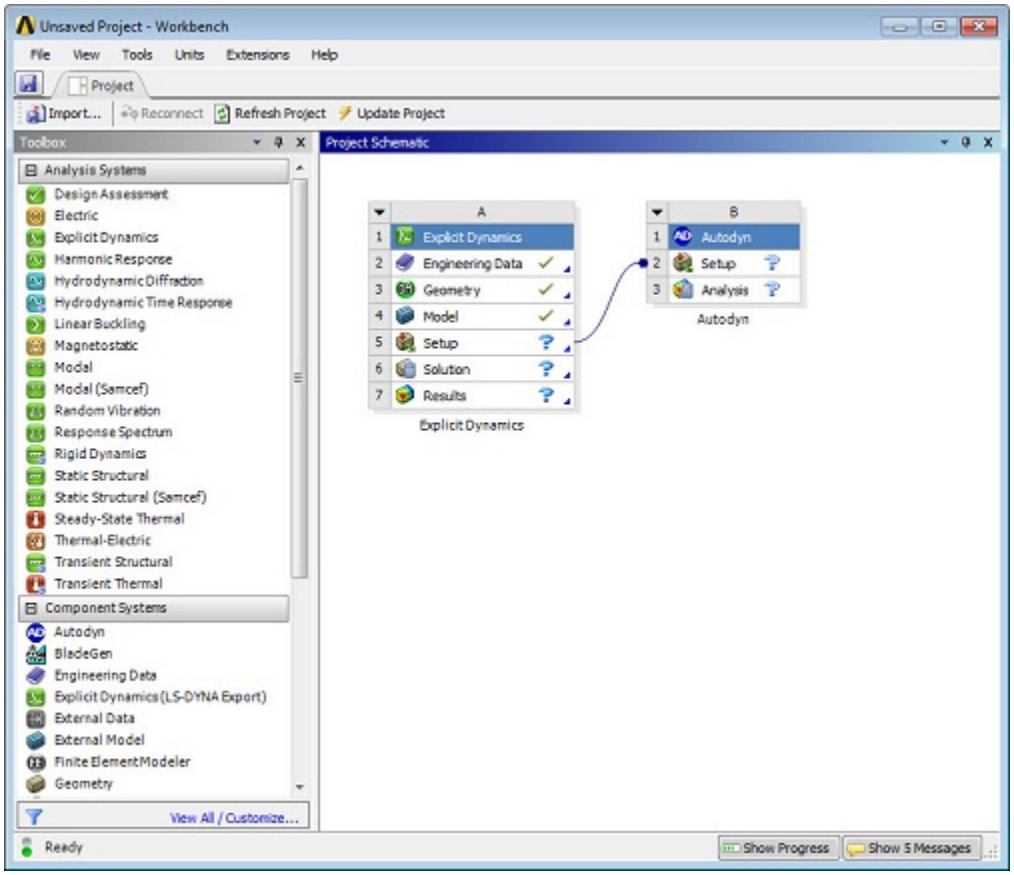
You can enter up to 256 characters.

1.12. Autodyn in Workbench

An overview of how to access Autodyn in Workbench is provided below. Full guidance on how to use Workbench is provided in the Workbench User's Guide.

The Workbench user interface consists primarily of a Toolbox region, Project Schematic, toolbar, and menu bar. Depending on the analysis type and/or application or workspace, you may also see other windows, tables, charts, and so on.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



One method of creating a project in Workbench is to drag an item such as a component or analysis system from the Toolbox to the Project Schematic, or double-click on an item in the toolbox to initiate the default action. You can also use the context menus, accessible from a right-mouse click, for additional options. You will view your analysis systems—the components that make up your analysis—in the Project Schematic, including all connections and links between the systems. The individual applications in which you work will display separately from the Workbench user interface, but the results of the actions you take in the applications may be reflected in the Project Schematic.

Full details of how to use Workbench are provided in the Workbench User's Guide.

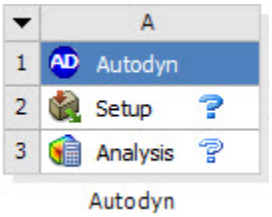
An Autodyn product license should enable three primary systems in the Toolbox of the Workbench user interface.

Autodyn System

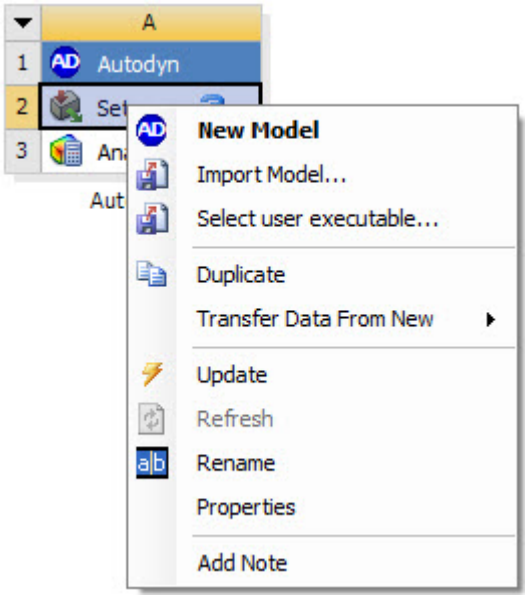
The Autodyn system is located in the Component Systems toolbox of the Workbench user interface. This system enables you to start the traditional Autodyn application and includes pre-processing, solve, and postprocessing options for all Autodyn solvers.

To start the Autodyn application, simply insert an Autodyn system into the project schematic and then double-click the Setup cell.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



The Autodyn application will start and you can set up, solve, and post-process models in the familiar Autodyn user interface. The default start up mode will be the generation of a new model. The start-up mode can be changed by a right-mouse click the Setup cell, after which you can select:



New Model

Default

Import Model

Open existing Autodyn Save file

Select user executable

Define where a user-customized version of Autodyn resides.

Duplicate

Make a copy of the model file.

Transfer Data From

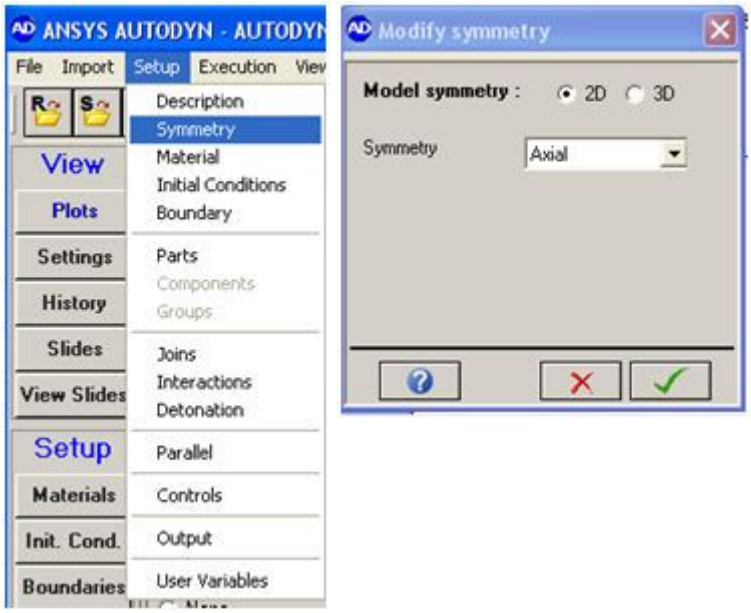
Model data from either the Mesh system or the Explicit Dynamics system are imported into Autodyn.

After selection of the desired Setup cell setting, double-clicking the Setup cell again will start Autodyn in the required mode.

Note:

- Model Name: The database for an Autodyn model will now always be admodel_0.ad. The project name is the now unique identifier for all databases (see below).
- 2D/3D symmetry: By default, the Autodyn application assumes you are performing a 3D simulation in the default unit set (mm, mg, ms). To define 2D axial or planar symmetry, use the **Setup** > **Symmetry** option in the toolbar after starting the Autodyn application.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Note:

- Saving a project: To save a model, the recommended approach is to save the Workbench project either from within the Autodyn application (File, Save Project) or from the project page of Workbench. This ensures all data used in the Autodyn Simulation is saved to disk.
- Exporting an Autodyn database: The Autodyn database (restart file) can be exported to a file (independent of the Workbench project) using the File, Export options in the Autodyn application.

Refer to Autodyn in the *Workbench User's Guide* for further details on the operations and properties available for an Autodyn system.

Explicit Dynamics system

The system contains a number of unique features for explicit dynamic simulation inside Workbench:

- Unidirectional and bidirectional links to CAD geometry
- Links to DesignModeler
- Advanced meshing capabilities tailored for Explicit Dynamics analysis
- Mesh independent loading and boundary conditions
- Fully parameterized engineering data, geometry, initial conditions, loads, boundary conditions
- Automated "What If" analysis capability
- Links to design exploration tools.
- Expressions are supported for Explicit, Explicit parallel, and Autodyn, but not Autodyn parallel analyses

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

The full capabilities of the Explicit Dynamics system are described in Explicit Dynamics Analysis Guide in the *Explicit Dynamics Analysis Guide*.

Models set up in the Explicit Dynamics system can be persistently linked to an Autodyn system to transfer material data, mesh, connections, loads and constraints, analysis settings, and so on into Autodyn.

For modeling complex geometries in Ansys Autodyn, we recommend you define the FE components of the model in an Explicit Dynamics system then transfer the model setup to the Autodyn system and add on your Euler-SPH regions and explicit FSI/Coupling as required.

Mesh System

The Mesh system is located in the Component Systems toolbox of the Workbench user interface. This system can be linked to either an Autodyn or Explicit Dynamics system and enables you to link to your model geometry and generate a mesh. The mesh can then be transferred to an Autodyn or Explicit Dynamics system for subsequent analysis.

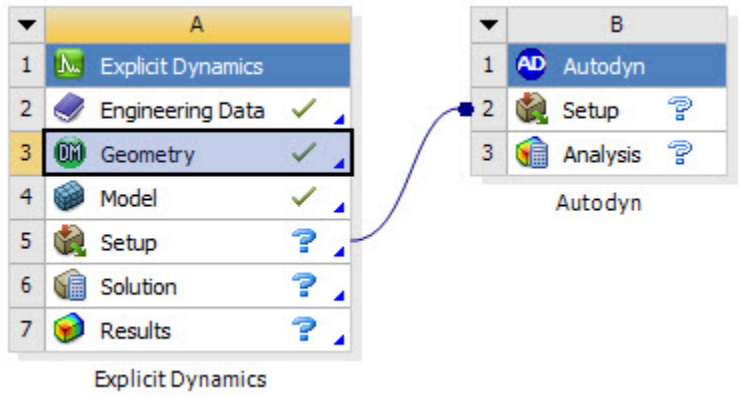
This system enables you to use a workflow to link Autodyn models to CAD/geometry and meshing tools. However the link is not persistent, which means that with every update of the workflow system, the definitions made in Autodyn (material filling, boundary condition assignment, etc.) are lost and have to be re-applied.

Therefore we recommend using the Explicit Dynamics system to link to CAD/geometry and generate a mesh for use in Autodyn where loads, constraints and material assignments are maintained after any change to the geometry or mesh.

Linking different systems

If the mesh/geometry of the application to be solved in Autodyn cannot be made with the simple model building tools available in the Autodyn application, we recommend the following approaches to build the Autodyn model.

Explicit Dynamics => Autodyn



The recommended approach for bringing Lagrangian FE based models with complex geometries into Autodyn is to start off with an Explicit Dynamics system.

- Define material data using the Engineering Data application

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- Link directly to CAD Geometry or use DesignModeler to generate geometry directly or adapt CAD geometry for simulation purposes
- Mesh the geometry and define connections (Contact, Joins, Spotwelds)
- Assign initial velocities to components
- Apply loads and constraints directly to the geometry
- Define solution controls and outputs
- Solve and Post process results

To transfer the model from an Explicit Dynamics system to the Autodyn application, link the Setup cell of the Explicit Dynamics system to the Setup cell of the Autodyn system. Open the Autodyn application by double-clicking the Autodyn system Setup cell. All relevant model information defined in Engineering Data, Geometry, Model and Setup components of an Explicit Dynamics system are transferred into the Autodyn application.

Once in the Autodyn application, you can complete the model setup (for example, add Euler and SPH regions) then solve in the Autodyn application.

The link generated between the Autodyn and Explicit Dynamics system is persistent. You can make changes to the model defined in the Explicit Dynamics system, update the Explicit Dynamics system Setup cell, then update the Autodyn system Setup cell. The latest model definition will be transferred into the Autodyn model while retaining additional features present only in the Autodyn application. This makes it very easy to make changes to model.

Note:

- Changes made to the model in the Autodyn application are not transferred back to the Explicit Dynamics system. The link is one-way.
- To avoid overwriting changes you make in the Autodyn application, we recommend that if a portion of the model is defined in the Explicit Dynamics system (for example and initial velocity) then only make changes to this in the Explicit Dynamics system. Transfer to the Autodyn model using the update procedure described above.
- The following functionality is transferred to and used in the Autodyn simulation; however it cannot be edited or reviewed in the Autodyn application:
  - Local coordinate systems.
  - Loading defined in tabular form.
  - General velocity boundary conditions.
  - Acceleration boundary conditions.
  - Joint definitions and their conditions.
  - Pointmass definitions.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

– Multiple load steps and the corresponding activation/deactivation of loads.

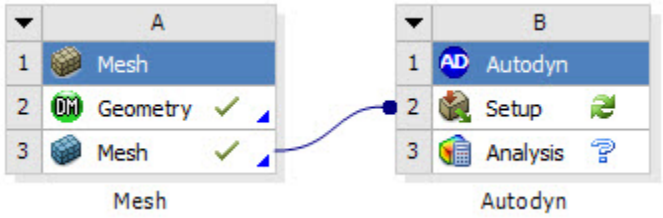
– Static Damping coefficient.

Note:

The step aware Static Damping coefficient can still be overruled by a global value under Solution Controls by changing the large value to a smaller value.

- An increase of the time limit in the Autodyn application will respect the load activation/de-activation of the last load step.
- Accelerations and Standard Earth Gravity are passed from the Explicit Dynamics system to the Autodyn application by means of acceleration boundary conditions. In the Autodyn application gravity may be added under Solution Controls, which will be added to the acceleration boundary conditions from the Explicit Dynamics system.
- Displacement boundary conditions defined in the Explicit Dynamics system are converted to equivalent velocity boundary conditions in the Autodyn application.
- Spot weld breaking criteria defined in the Explicit Dynamics system will be converted from a stress to a force criterion
- This link will also transfer parallel processing settings; however the linked Autodyn component system will not directly solve in parallel because the Autodyn system has not started the proper number of worker executables yet. In order for the Autodyn solver to run in parallel, the model (zero) file can be used as input for a parallel solve via the appropriate command line instructions which are described in Establishing a Parallel Processing Environment in the *Autodyn Parallel Processing Guide*. Alternatively, the parallel settings can be deselected in the component system in order to return to a serial simulation.
- Detonation Points transferred from Explicit Dynamics will be updated to reflect changes in the Explicit Dynamics system. Any Detonation Points defined in Autodyn will be lost after updating a linked system.

Meshing => Autodyn



If you only want to transfer a Mesh (of complex geometry) into Autodyn, then this can be achieved using a Mesh system. Start by inserting a Mesh system from the Components toolbox.

- Link directly to CAD Geometry or use DesignModeler to generate geometry directly or adapt CAD geometry for simulation purposes

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- Mesh the geometry
- Create Named Selections

Note:

When you open the Meshing application, you must set the Physics Preference for the mesh to Explicit. In the Meshing application, you can set the **Physics Preference** in the wizard that initially comes up when you open the Meshing application, or by clicking on the Mesh object in the tree and changing the **Physics Preference** field in the Details panel.

To transfer a mesh from the Mesh to the Autodyn application, link the Mesh cell of the Mesh system to the Setup cell of the Autodyn system. Open the Autodyn application by double-clicking the Autodyn system Setup cell. The mesh and Named selections will be transferred to Autodyn.

The link generated between the Autodyn and Mesh systems is not persistent. This means that with every update of the workflow system the definitions made in Autodyn (material filling, boundary condition assignment, and so on) are lost and have to be re-applied.

Therefore, we recommend using the Explicit Dynamics system to link to CAD/geometry and generate a mesh for use in Autodyn, where loads, constraints and material assignments are maintained after any change to the geometry or mesh.

Note:

- Connections defined in the Mesh system will not be transferred to the Autodyn application.
- Material assignments, Boundary Conditions and Loads defined in the Autodyn application will be lost during the update and will have to be re-applied.

Data Transferred by linking different systems to Autodyn

Node Coordinates

The nodal coordinates are transferred.

Mesh Elements

The mesh elements are transferred, noting that Pyramid elements are not supported within Autodyn, so these are converted into two tetrahedra.

Body Definitions

The definitions of bodies in the Workbench are imported as Parts in Autodyn.

In addition, for Meshing, Simulation and Finite Element Model Objects, the following are transferred.

Parts Definitions

The definitions of parts (both multi body parts and single body parts) are shown as Components in Autodyn.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Named Selection

The named selections are represented as Groups in Autodyn. Vertex and Edge Named Selections are transferred as nodal Groups. Surface Named Selections are transferred as a nodal Group and a face Group. Body Named Selections are transferred as a nodal Group and an elemental Group.

Unit Setup

The units of length are transferred into Autodyn and the units for mass and time are set up as shown in the following table.

Length	Mass	Time
um	pg	ms
mm	mg	ms
cm	gm	us
m	kg	s
in	lbm	ms
ft	lbm	ms

Material Associations

Each Body (Autodyn Part) transferred is assigned a separate material.

Shell thickness

Shell parts will be assigned with the thickness as defined in the Workbench object. An initial condition will be created for each part.

Beam Cross-Sections and Orientations

The beam cross sections associated to elements and their orientations are transferred and applied correspondingly.

In order to update the model in Autodyn to that in the currently selected project page item, make sure that the currently selected cycle is 0 then use the **Update Model from Workbench** option, found in the **Import** menu or on the toolbar. This function will update the parts that are in Autodyn to reflect the mesh in Workbench. If a part contains a single material, then the updated parts will be refilled with any materials and initial conditions that are associated. New parts are added to the model as appropriate; deleted parts should be manually removed from the Autodyn model.

1.13. Autodyn Solver Options in Workbench

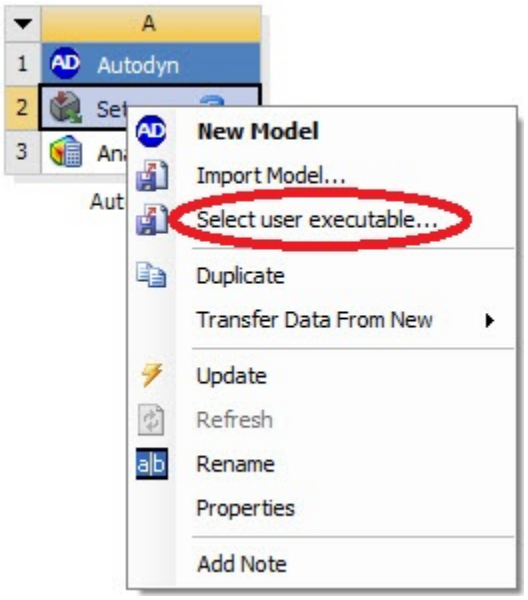
There are a number of Autodyn solver options, which may be changed by a right-mouse click the Setup cell of the Autodyn system.

User Executable Location

By default, Workbench will start the standard Autodyn executable. You may also choose an executable created with a user-subroutine. To run your own user executable from the project schematic, right

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

click the Setup cell of the Autodyn system and left click **Select user executable**, after which you will be asked for the user executable name and location.



Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information
of ANSYS, Inc. and its subsidiaries and affiliates.

Chapter 2: Autodyn Pull Down Menus

Pull-down menus let you access all the features and dialog panels/windows in Autodyn.

Most options can also be accessed through buttons on the toolbar and navigation bar.

Options that are accessible only through pull-down menus are shown below in **bold**.

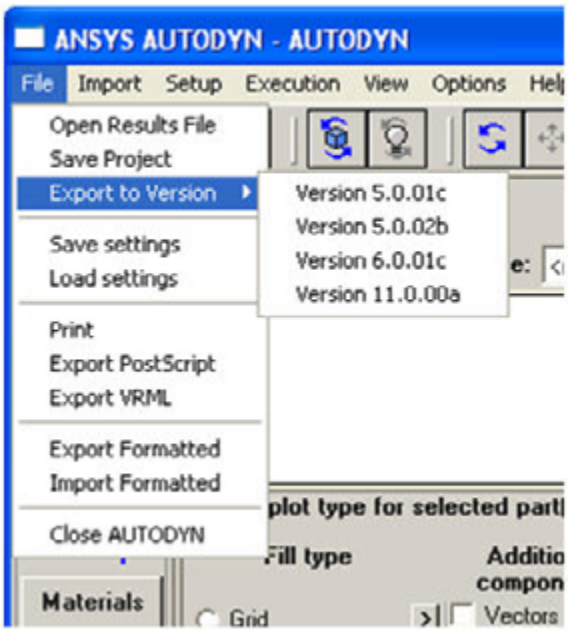
Options that are available only when Autodyn is started outside the Workbench environment are shown below in *italics*.

FILE	IMPORT	SETUP	EXECUTION	VIEW	OPTIONS	HELP
New *Open* Open Results File *Save* **Save Project** *Save As* *Save As Version* **Export to Version** *Manage Folders* Save settings Load settings Print	**from TrueGrid® (.zon)** **from ICEMCFD (.geo)** **from LS-DYNA (.k)** **from MSC.NASTRAN bdr (.dat)** **Convert IJK to unstructured**	**Description** **Symmetry** Material Initial Conditions Boundary Parts Components Groups Joins Interactions Detonation Parallel Controls Output User Variables	Run Interactively **Batch Runner**	Plots Plot Settings History Graphs Slideshow **Performance Profiler**	**Interactivity** **Clipping Planes** **Default Settings** **Pause License** **Create User-Subroutine Project**	**Contents** **About**

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

FILE	IMPORT	SETUP	EXECUTION	VIEW	OPTIONS	HELP
Export PostScript **Export VRML** *Save Formatted* **Export Formatted** *Open Formatted* **Import Formatted**						

2.1. Pull-Down Menu - File



New

Creates a new model

Open

Opens (loads) an existing model saved in a binary file.

Open Results Files

Opens an existing results file. For more details of results files, see the section Outputs - Results Files.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Save

Saves the current model using the current Ident for the filename to a binary file.

Save Project

Saves the Workbench project and ensures that the Autodyn analysis model and all Workbench related data is saved to disk.

Save As

Saves the current model using a new Ident for the filename.

Save As Version

Saves the current model in a format supported by a previous version.

Export to Version

Saves the current model outside the Workbench project in a format supported by a previous version.

Manage Folders

Enables you to manage your project folders.

Save settings

Saves the settings for the current plot in a settings file.

Load settings

Loads and applies plot settings from a saved settings file.

Print

Outputs data for the current plot to a printer.

Export PostScript

Outputs data for the current plot to a PostScript file.

Export VRML

Outputs data for the current plot to a VRML file.

Save Formatted

Saves the current model using the current Ident for the filename to a formatted (ASCII) file.

Export Formatted

Saves the current model to a formatted (ASCII) file.

Open Formatted

Opens (loads) an existing model saved in a formatted (ASCII) file.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Import Formatted

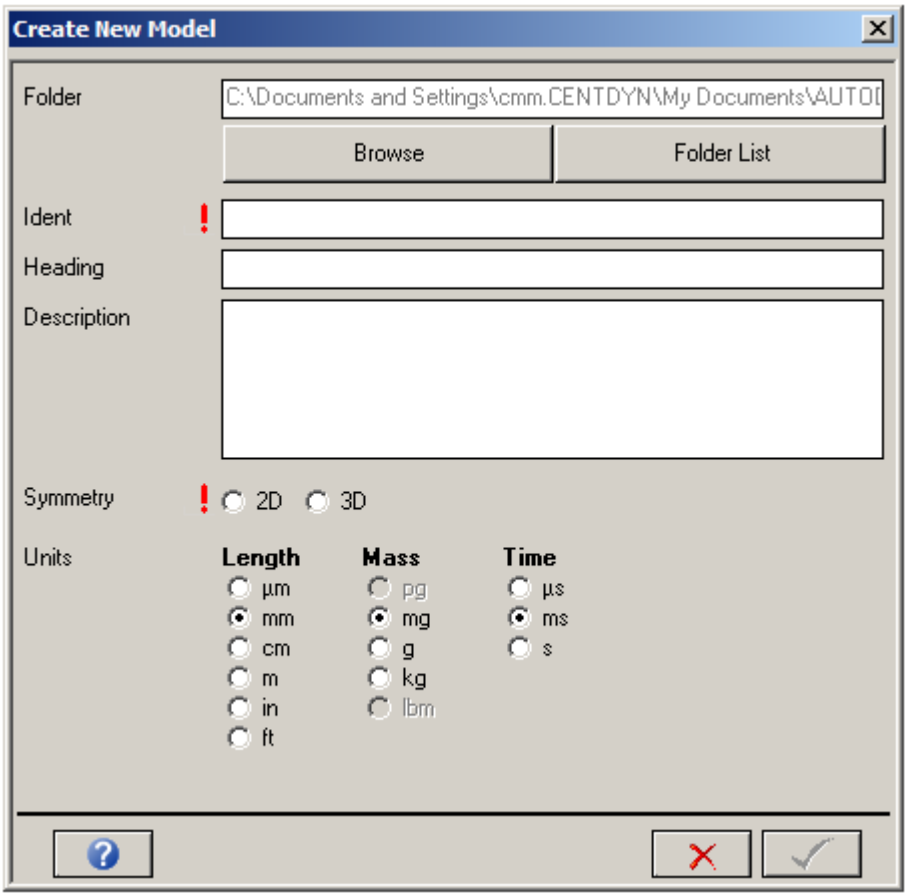
Lets you open (load) an existing model saved in a formatted (ASCII) file.

Close Autodyn

Exits Autodyn.

2.1.1. Pull-Down Menu - File - New

This menu is available only when Autodyn is started outside Workbench.



Starts the creation of a new model.

Folder

Shows the folder where files for the model will be kept.

Click the **Browse** button to select a different folder.

Click the **Folder List** button to select a folder from your current list of folders or manage your folder list

Ident

Enter an Ident for your model in this field.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

The Ident you choose is used to identify all the files created for your model.

For example, If your Ident is **MyImpactProblem**, then the cycles zero Save file for the problem will be named **MyImpactProblem_0.ad**.

Heading

Enter a heading for your model in this field.

Headings appear in plots and other output for your model.

Description

Enter a more detailed description of your model here if you want.

Symmetry

Select the dimension (2D or 3D) and symmetry you want to use for your model.

- In 2D you must choose either **Axial** or **Planar** symmetry
- In 3D you do not need to choose a symmetry, but you can optionally select **X**, **Y** and/or **Z** symmetry.

  (For example, if you select X symmetry, then the plane X=0 is assumed to be a symmetry plane).

Units

Select the units of length, mass and time you want to use for your model.

The default units (mm, mg, ms) will give good results, with small rounding errors, for most models.

US Customary Units

US customary units have been added to units options when creating a new model.

These are:

- Length unit (inch or foot)
- Mass unit (pound)

These units can be written in short as "in", "ft", and "lb". One of the three time units can be used in conjunction with the US customary units: microsecond, millisecond, or second.

Note they cannot be used together with Metric Units. For example, you cannot use inch for length unit and kilogram for the mass unit.

The following table shows the unit for each type of physical variables in the US customary system.

	in, lb, s	**in, lb, ms**	**in, lb, µs**	**ft, lb, s**	**ft, lb, ms**	**ft, lb, µs**
Length	in	in	in	ft	ft	ft
Mass	lb	lb	lb	lb	lb	lb
Time	s	ms	µs	s	ms	µs

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

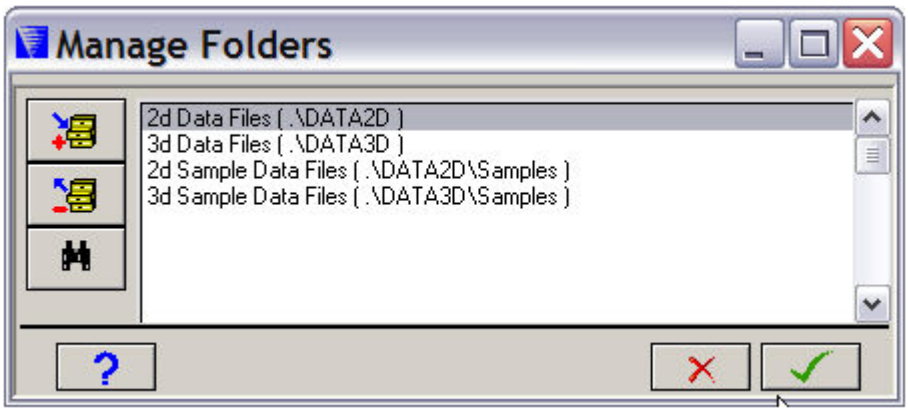
	in, lb, s	in, lb, ms	in, lb, µs	ft, lb, s	ft, lb, ms	ft, lb, µs
Temperature	K	K	K	K	K	K
Density	$lb.s^2/in^4$	$lb.ms^2/in^4$	$lb.\mu s^2/in^4$	$lb.s^2/ft^4$	$lb.ms^2/ft^4$	$lb.\mu s^2/ft^4$
Inertia	$lb.s^2.in$	$lb.ms^2.in$	$lb.\mu s^2.in$	$lb.s^2.ft$	$lb.ms^2.ft$	$lb.\mu s^2.ft$
Volume	in^3	in^3	in^3	ft^3	ft^3	ft^3
Force	lb	lb	lb	lb	lb	lb
Pressure	psi	psi	psi	psf	psf	psf
Impulse	lb.s	lb.ms	lb.µs	lb.s	lb.ms	lb.µs
Velocity	in/s	in/ms	in/µs	ft/s	ft/ms	ft/µs
Work	lb.in	lb.in	lb.in	lb.ft	lb.ft	lb.ft
Strain Rate	1/s	1/ms	1/µs	1/s	1/ms	1/µs
Energy	lb.in	lb.in	lb.in	lb.ft	lb.ft	lb.ft
Energy/Unit Mass	in^2/s^2	in^2/ms^2	$in^2/\mu s^2$	ft^2/s^2	ft^2/ms^2	$ft^2/\mu s^2$
Energy/Unit Volume	lb/in^2	lb/in^2	lb/in^2	lb/ft^2	lb/ft^2	lb/ft^2
Fracture Energy	lb.s	lb.ms	lb.µs	lb.s	lb.ms	lb.µs
Fracture Toughness Squared	lb^2/in^2	lb^2/in^2	lb^2/in^2	lb^2/ft^2	lb^2/ft^2	lb^2/ft^2
Specific Head Capacity	$in^2/s^2.K$	$in^2/ms^2.K$	$in^2/\mu s^2.K$	$ft^2/s^2.K$	$ft^2/ms^2.K$	$ft^2/\mu s^2.K$
Thermal Conductivity	lb.s/K	lb.ms/K	lb.µs/K	lb.s/K	lb.ms/K	lb.µs/K

Micron Unit System

A new micron (µm) length system has been introduced. This can be used in conjunction with a mass unit of picogram (pg) and time unit of microsecond or millisecond.

2.1.2. Pull-Down Menu - File - Manage Folders

This menu is available only when Autodyn is started outside Workbench.



This window lets you manage a list of the folders containing your Autodyn models.

You can add and delete folders from this list as your projects change.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Folder List

The scroll box contains a list of the current folders you have selected. For each folder you see its alias (if defined) followed by its full path name in parentheses.

Double clicking on any folder in this list selects it as the directory for your current model.

Select / Create Folder

Click this button to add another folder to your list (using a browser). After selecting the directory for the folder, you can optionally specify an alias for the folder.

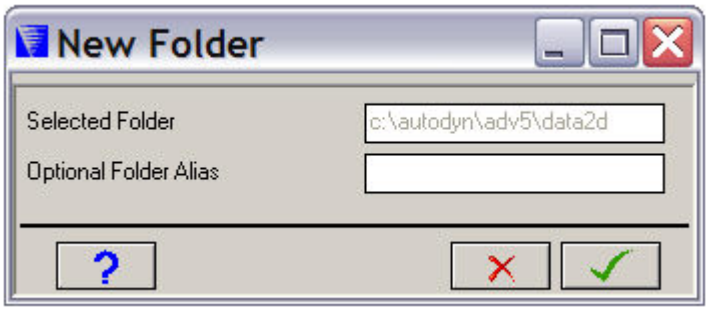
Remove Folder from list

Click this button to remove the selected folder from your list.

View Folder Contents

Click this button to view the model files in the selected folder.

2.1.2.1. Pull-Down Menu - File - Manage Folders - New



This window lets you define an optional alias name for a folder.

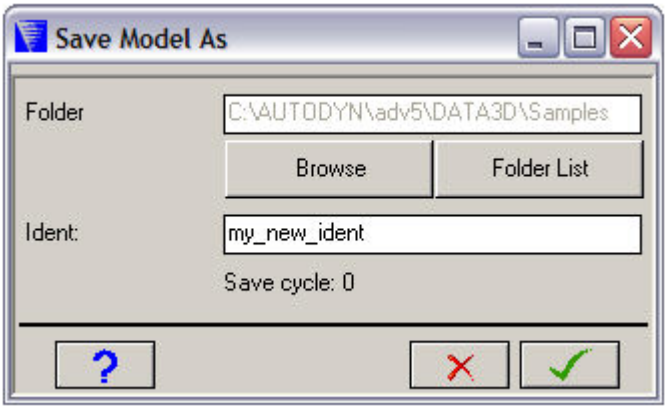
Enter the alias name in the field provided.

If you leave this field blank, the full folder name will be used to reference the folder.

2.1.3. Pull-Down Menu - File - Save As

This menu is available only when Autodyn is started outside Workbench.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



This window lets you save the current model under a new Ident.

Folder

The current folder in which model files are saved.

Browse

Browse to a new folder for saving model files.

Folder List

Select a folder from your folder list for saving model files and/or manage your folder list.

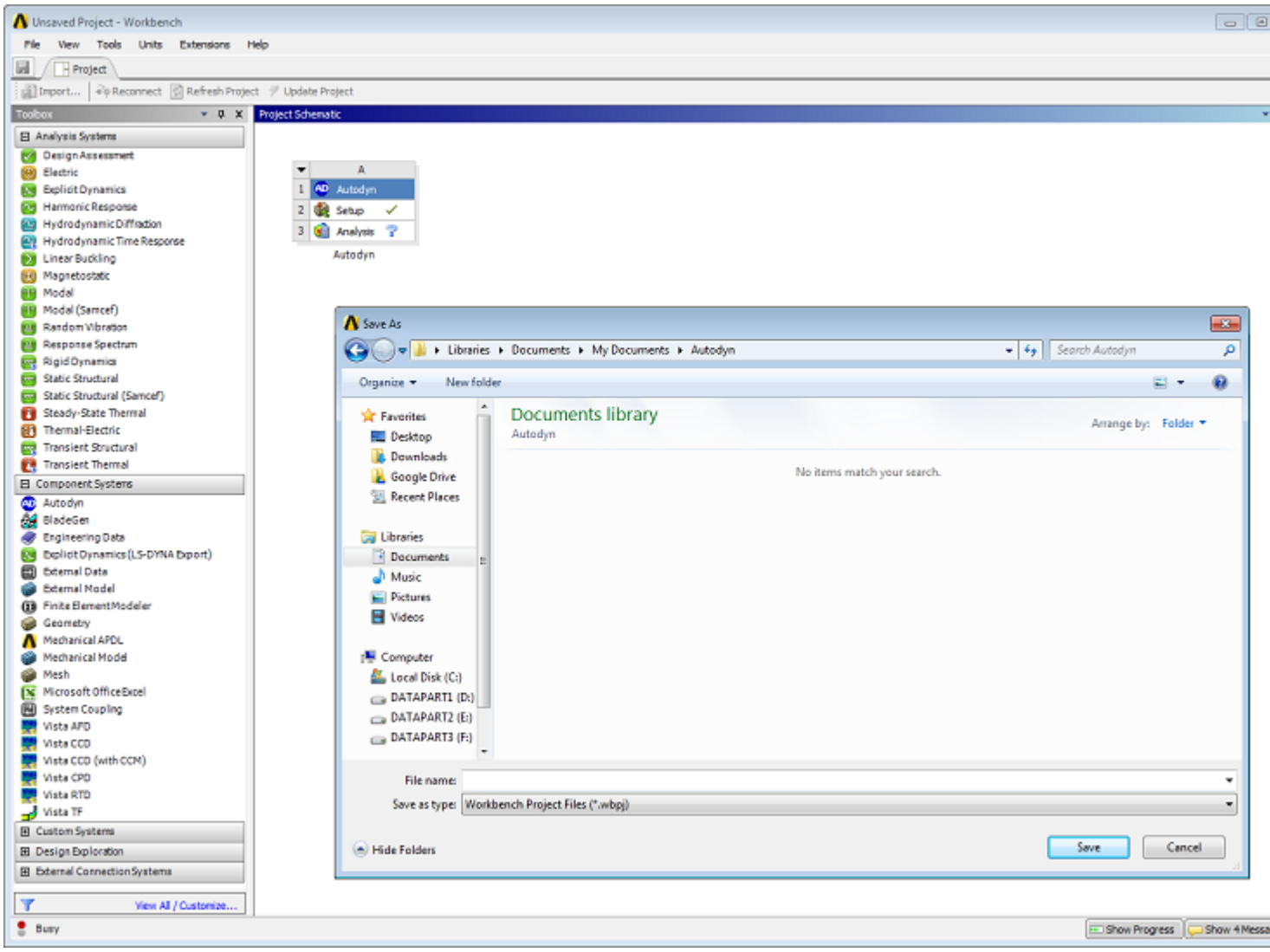
Ident

The new Ident you want to use for saving model data.

2.1.4. Pull-Down Menu - File - Save Project

This option brings up the Workbench project page and allows you to save the current Workbench project and ensures that the Autodyn analysis model and all Workbench related data is saved to disk.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Save in

The folder in which the Workbench project tree will be saved; you can select a different folder if required.

File Name

The user-defined project name.

2.1.5. Pull-Down Menu - File - Save As Version

This menu is available only when Autodyn is started outside Workbench.

This option lets you create an Autodyn save file in a format supported by a previous version. The formats available are listed and the desired format should be selected from the list.

As features and options available in the current version, but not in the version selected, will be lost, we recommend that you always choose a new filename when using this option.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

2.1.6. Pull-Down Menu - File - Export to Version

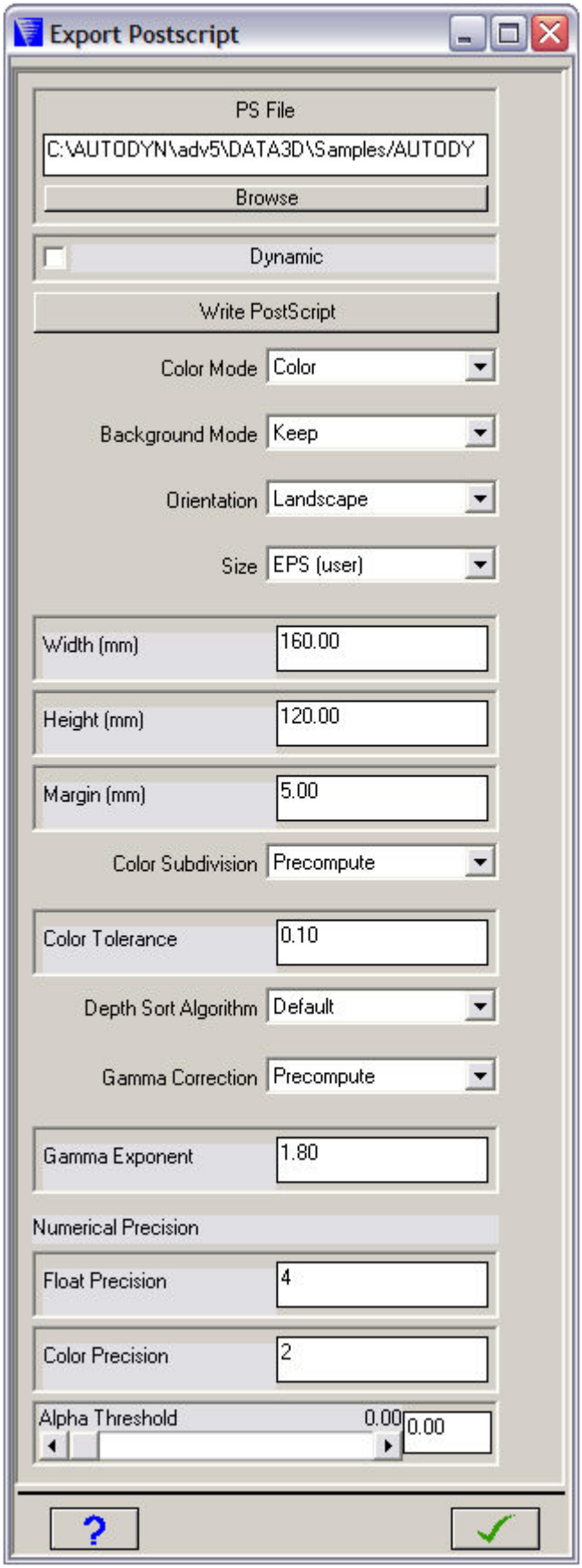
This option lets you create an Autodyn save file in a format supported by a previous version. The formats available are listed.

As features and options available in the current version, but not in the version selected, will be lost, we recommend that you always choose a new filename when using this option.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

2.1.7. Pull-Down Menu - File - Export PostScript

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

This window lets you output the current plot to a PostScript file.

PS file

Enter the name of the PostScript file in this field (use the Browse button to navigate to a directory/file if you wish).

Dynamic

Check this box if you want a new PostScript file to be output each time the input view is rendered. Otherwise, you must explicitly request a new file by clicking the **Write PostScript** button.

Write PostScript

If **Dynamic** is not checked, click this button to generate a PostScript output file.

Color Mode

Sets the color format used in the output file.

Color

Each color is specified as an RGB triple in the output file. The output file can be rendered in color on a color printer, or in greyscale on a monochrome printer.

Greyscale

Each color is converted to a single intensity value. The output file renders only in greyscale on both color or monochrome printers. Greyscale files are smaller than those written with full color information.

Background Mode

Sets how the background of the scene will be rendered.

White

Force the scene to be rendered on a white background. This typically means that the printer does not apply any color or shading to the background.

Black

Force the scene to be rendered on a black background.

Keep

Use the view background color specified in the AVS/Exclick viewer as the PostScript background color.

Orientation

Enables you to select the page layout.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Landscape

The x-axis of the scene device coordinates is oriented along the longer side of the paper.

Portrait

The x-axis of the scene device coordinates is oriented along the shorter side of the paper.

Size

Enables you to select the paper size for the output.

EPS (user)

Encapsulated PostScript format, where you can specify the dimensions of the output, and the resulting file can be embedded in other documents. You must set the width, height and margin in the fields provided if you select this option.

A (letter)

Standard US letter paper size (280mm x 216mm)

B

Ledger paper size (432mm x 280mm).

A4

Standard European A4 paper size (297mm x 210mm).

A3

A3 paper size (420mm x 297mm).

A0

A0 paper size (1189mm x 841mm).

Width, Height, Margin

Use these fields to specify the width and height and margin of EPS output.

Color Subdivision

Enables you to select how the color is interpolated for polylines and surfaces, in 2D and 3D, with per-vertex color or lighting of 3D surfaces. When subdivision is enabled (**Precompute** or **Postcompute**), the subdivision proceeds until a color tolerance is reached. This adaptive method gives the best quality results without subdividing primitives with small color gradients.

None (flat)

No color interpolation is performed. The output polyline and triangle primitives are drawn with constant color per-segment, or per-facet. Incoming vertex colors are averaged to produce the segment or facet color, where necessary.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Precompute

The OutputVPS module subdivides line segments and triangles that have per-vertex color from data mapping, or as the result of lighting 3D surfaces. The subdivision generates many smaller primitives with linearly interpolated color. Precomputing the subdivision increases the output file size, and the subdivision cannot be changed during printing.

Postcompute

Special interpolation macros are written to the output file, together with the color tolerance parameter. Primitives are written with color per-vertex information in the PostScript output. The subdivision is made at print time, in the PostScript processor. This gives high quality output, with small file size, at the expense of slower printing. The color tolerance parameter can be edited by hand to change the color resolution after the file has been generated.

Color tolerance

Use this field to specify the absolute difference in color space, for the color subdivision algorithms: **Precompute** and **Postcompute**. In Postcompute mode, the color tolerance is written to the output file. In precompute mode it is used by the OutputVPS module to subdivide the line segments and triangle facets which have per-vertex color. This real value must be in the range 0.0 to 1.0. Smaller values signify a finer subdivision. Values close to zero will force a very large number of primitives to be generated, either precomputed in the output file (large file, slow printing), or postcomputed internally within the printer (slow printing). A value of 1.0 will not cause any subdivision to occur, even when it is enabled by the subdivision flag. The color tolerance user interface is not displayed for color subdivision None.

Depth Sort Algorithm

Select the depth sort algorithm you want to use from the pull-down menu.

Default

A simple sort by the minimum z coordinate of each primitive object. While this is well suited for many applications, it can result in visual artifacts in complex scenes.

NNS

The Newell and Sancha hidden surface removal algorithm. When a possible visual artifact is detected, the algorithm first tries to interchange the conflicting primitives. If there is still a conflict (consider the case of intersecting triangles), then one of the primitives is split along the line(s) of intersection.

Gamma Correction

Enables you to select how final colors are modified to take account of nonlinear effects in the printing device, or perhaps to match the nonlinear effects of the same scene displayed on a raster monitor.

None

No gamma correction is performed. Color values are assumed to behave linearly.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Precompute

The OutputVPS module modifies all color values (gray, red, green, blue), before writing the values to the file. Thus precomputed gamma correction is applied after precomputed color subdivision, but before postcomputed color subdivision. This means that postcomputed linear interpolation may not give the expected appearance. Precomputed gamma marginally slows file generation, but does not affect file size. The precomputed gamma factor cannot be changed after the file has been generated.

Postcompute (default)

The gamma exponent is written to the output file, and an exponential function is set for color transfer in the printer. Each rendered color is then gamma corrected at print time. Postcomputing the gamma correction increases the file size by only a few bytes, but it will slow down printing. The gamma factor can be edited by hand after the file has been generated.

Gamma exponent

Use this field to specify the gamma factor to be applied to color components, either precomputed in the output file, or postcomputed by the printer during printing. The gamma factor is an exponential factor applied to each color component independently (gray, red, green, blue). Its value must be greater than 0.0. A value of 1.0 leaves color unchanged, values between 0.0 and 1.0 darken the colors, and values greater than 1.0 lighten (desaturate) the colors. The default value is 1.8.

Numerical Precision

Float Precision

Use this field to specify the floating point precision used to write real values to the output file. The values include spatial coordinates and transformations, but not colors. The value must be in the range 1 to 8. The default value is 4.

Color Precision

Use this field to specify the floating point precision used to write color values to the output file. The color values can be gray, red, green or blue. The value must be in the range 1 to 8. The default value is 2.

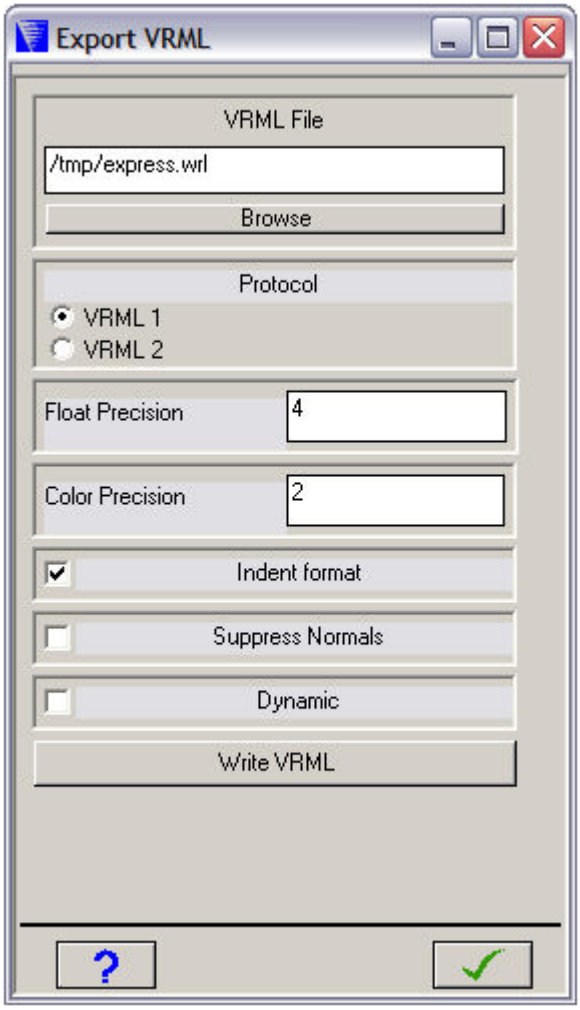
Alpha Threshold

Sets the Alpha threshold.

2.1.8. Pull-Down Menu - File - Export VRML

This option brings up the Workbench project page, allows you to save the current Workbench project and ensures that the Autodyn analysis model and all Workbench related data is saved to disk.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



This window lets you output the current plot to a VRML file.

VRML file

The name of the VRML file (use the Browse button to navigate to a directory/file if you wish).

Protocol

The VRML protocol you want to use to write the output file:

VRML1

A clear text ASCII encoding of the VRML 1.0c standard (29 January 1996).

VRML2

A clear text UTF8 encoding of the VRML 2.0 standard, ISO/IEC CD 14772 (4 August 1996).

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Float Precision

Specifies the floating point precision when real values are written to the output file. The values include spatial coordinates, normal vectors, texture coordinates, material coefficients, and transformations, but not colors. The value must be in the range 1 to 8. The default value is 4.

Color Precision

Specifies the floating point precision when real values are written to the output file. The values include diffuse material colors, emissive material colors, and light colors. The value must be in the range 1 to 8. The default value is 2.

Indent format

Controls whether the file to be written with indented nesting of VRML nodes. Indentation improves legibility, but increases the size of the output file.

Supclick Normals

Controls whether normals are written to the VRML.

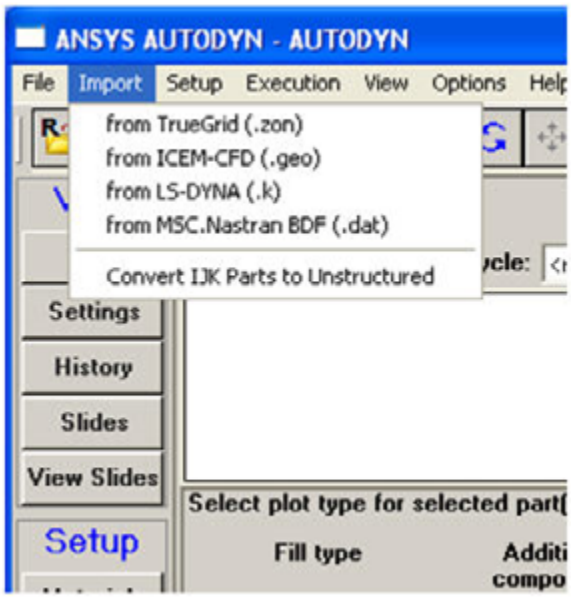
Dynamic

Controls whether a new VRML file is output each time the input view is rendered. Otherwise, you must explicitly request a new file by clicking the **Write VRML** button.

Write VRML

When the dynamic switch is FALSE, this one-shot parameter triggers the generation of a VRML output file. When the dynamic switch is TRUE, the user interface for this parameter is removed.

2.2. Pull-Down Menu - Import



Enables you to import grid data from other programs.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

from True*Grid*® (.zon)

Imports block zoning data from True*Grid*®.

from ICEM (.geo)

Imports block zoning data from ICEM-CFD.

from LS-DYNA (.k)

Imports .k files from LS-DYNA.

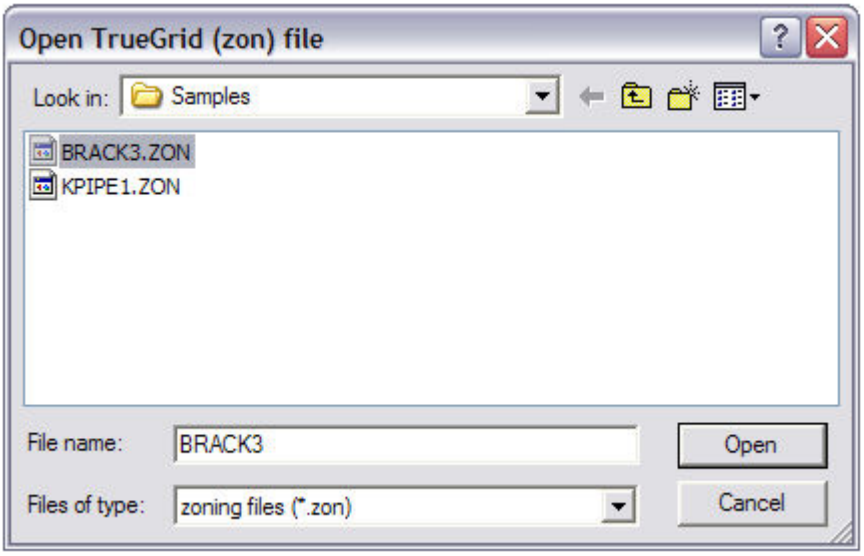
from MSC.NASTRAN bdf (.dat)

Imports .bdf files from Nastran.

Convert IJK to Unstructured

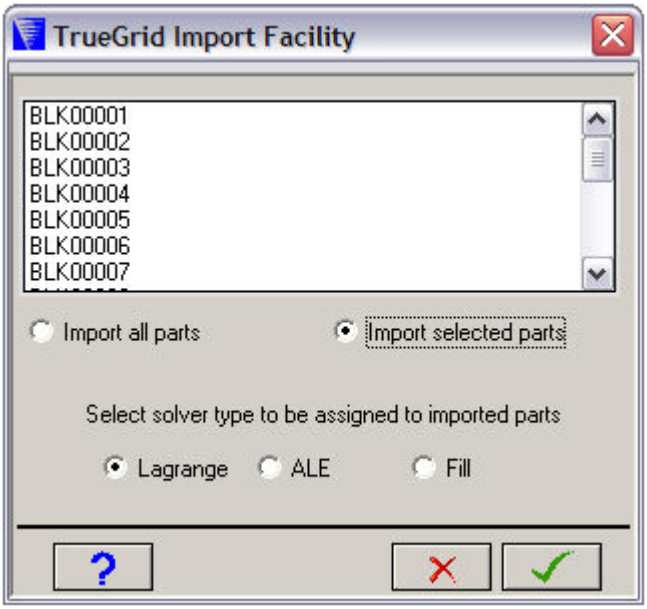
Converts existing IJK structured Parts to Unstructured Parts. This option is available for 3D analyses only.

2.2.1. Pull-Down Menu - Import - True*Grid*®



This window lets you import grid data from True*Grid*®.

Navigate to the True*Grid*® zoning file you want to import (**.zon**) and click Open to open the following window.



Blocks

The True*Grid*® zoning file contains one or more blocks of nodes (imported as Parts). These are displayed in the scrolling box at the top of the window.

If Import selected parts is checked, you can select one or more blocks.

Import all Parts

Imports all the blocks as Parts.

Import selected Parts

Imports selected blocks as Parts. Select the blocks you want to import in the top box.

Select solver type

Select the solver type you want to assign to the imported Parts.

Click to import the Parts.

Parts are imported with their block name. You can change these block names later in the Parts panel.

TrueGrid® zoning file format

There are two formats of True*Grid*® zoning files supported by Autodyn - implicit and explicit. The two formats are illustrated below by an example which generates a single cell cubic part (IMAX=2, JMAX=2, KMAX=2) called SUB1 with corners (0.0,0.0,0.0) and (1.0,1.0,1,0) and a four cell cuboid part (IMAX=2, JMAX=2, KMAX=3) called SUB2 with corners (2.0,2.0,2.0) and (3.0,3.0,4.0).

The implicit form of this data would be as follows:

```
IMPLICIT        Indicates implicit input is to follow
SUB1            Name of the part for which data follows
```


Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

```
1 2 1 2 1 2     The (I,J,K) range for the data I=1, 2, J=1,2, K=1,2
0.0 0.0 0.0     (x,y,z) coordinates for node with index (1,1,1)
0.0 0.0 1.0     (x,y,z) coordinates for node with index (1,1,2)
0.0 1.0 0.0     (x,y,z) coordinates for node with index (1,2,1)
0.0 1.0 1.0     (x,y,z) coordinates for node with index (1,2,2)
1.0 0.0 0.0     (x,y,z) coordinates for node with index (2,1,1)
1.0 0.0 1.0     (x,y,z) coordinates for node with index (2,1,2)
1.0 1.0 0.0     (x,y,z) coordinates for node with index (2,2,1)
1.0 1.0 1.0     (x,y,z) coordinates for node with index (2,2,2)
SUB2            Name of the part for which data follows
1 2 1 2 1 3     The (I, J, K) range for the data I=J, 2, J=1,2, K=1,3
2.0 2.0 2.0     (x,y,z) coordinates for node with index (1,1,1)
2.0 2.0 3.0     (x,y,z) coordinates for node with index (1,1,2)
2.0 2.0 4.0     (x,y,z) coordinates for node with index (1,1,3)
2.0 3.0 3.0     (x,y,z) coordinates for node with index (1,2,1)
2.0 3.0 4.0     (x,y,z) coordinates for node with index (1.2,2)
3.0 2.0 2.0     (x,y,z) coordinates for node with index (2,1,1)
3.0 2.0 3.0     (x,y,z) coordinates for node with index (2,1,2)
3.0 2.0 4.0     (x,y,z) coordinates for node with index (2,1,3)
3.0 3.0 2.0     (x,y,z) coordinates for node with index (2,2,1)
3.0 3.0 3.0     (x,y,z) coordinates for node with index (2.2,2)
3.0 3.0 4.0     (x,y,z) coordinates for node with index (2,2,3)
END
```

The explicit form of this data would be as follows:

```
EXPLICIT              Indicates explicit input is to follow
SUB1                  Name of the part for which data follows
1 1 1 0.0 0.0 0.0     (x,y,z) coordinates for (1,1,1)
1 1 2 0.0 0.0 1.0     (x,y,z) coordinates for (1,1,2)
1 2 1 0.0 1.0 0.0     (x,y,z) coordinates for (1,2,1)
1 2 2 0.0 1.0 1.0     (x,y,z) coordinates for (1,2,2)
2 1 1 1.0 0.0 0.0     (x,y,z) coordinates for (2,1,1)
2 1 2 1.0 0.0 1.0     (x,y,z) coordinates for (2,1,2)
2 2 1 1.0 1.0 0.0     (x,y,z) coordinates for (2,2,1)
2 2 2 1.0 1.0 1.0     (x,y,z) coordinates for (2,2,2)
SUB2                  Name of the part for which data follows
1 1 1 2.0 2.0 2.0     (x,y,z) coordinates for (1,1,1)
1 1 2 2.0 2.0 3.0     (x,y,z) coordinates for (1,1,2)
1 1 3 2.0 2.0 4.0     (x,y,z) coordinates for (1,1,3)
1 2 1 2.0 3.0 2.0     (x,y,z) coordinates for (1,2,1)
1 2 2 2.0 3.0 3.0     (x,y,z) coordinates for (1.2,2)
1 2 3 2.0 3.0 4.0     (x,y,z) coordinates for (1,2,3)
2 1 1 3.0 2.0 2.0     (x,y,z) coordinates for (2.1,1)
2 1 2 3.0 2.0 3.0     (x,y,z) coordinates for (2,1,2)
2 1 3 3.0 2.0 4.0     (x,y,z) coordinates for (2,1,3)
2 2 1 3.0 3.0 2.0     (x,y,z) coordinates for (2,2,1)
2 2 2 3.0 3.0 3.0     (x,y,z) coordinates for (2.2,2)
2 2 3 3.0 3.0 4.0     (x,y,z) coordinates for (2,2,3)
STOP                  Indicates end of all data
```

With the explicit format, it is not necessary to define coordinates for a complete block of nodes in index space, and the order in which nodes are defined is unimportant.

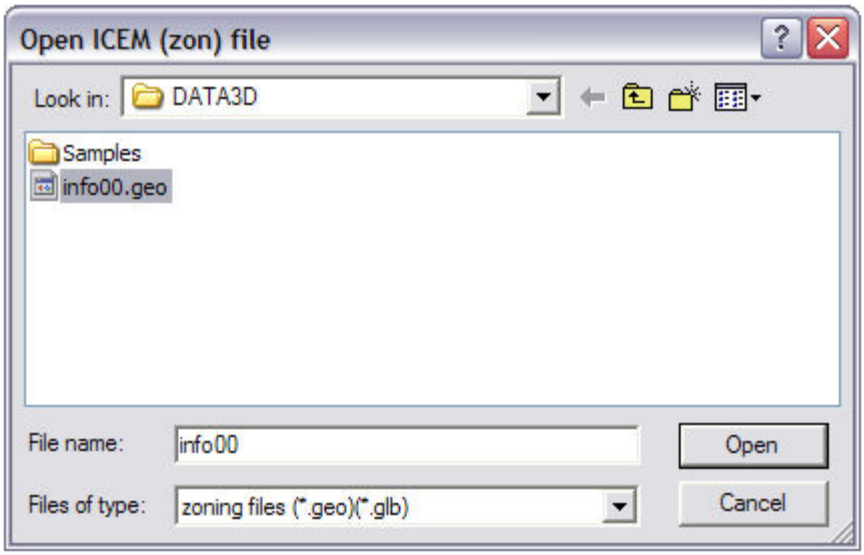
When this option is used for a specific block of nodes in index space, the "indent.zon" file is scanned and only those nodes defined in this file and within the specified part and block to be initialized will be assigned coordinate values. Thus, standard zoning options could be used to define the bulk of the grid and the "User" option called to redefine the coordinates of a few isolated nodes.

Before reading in an explicit zoning file, the parts must already be created. The file can then be read in using the Import button on the Parts Zoning panel.

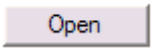
Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

The above example is for 3D. For 2D models the format is the same except of course there is no K index or Z coordinates.

2.2.2. Pull-Down Menu - Import - ICEM



This window lets you import grid data from ICEM-CFD.

Navigate to the ICEM-CFD zoning file you want to import (**.geo**) and click  to open the following window.

Domains

The ICEM-CFD zoning file contains one or more domains of nodes (imported as Parts). These are displayed in the scrolling box at the top of the window. If Import selected parts is checked, you can select one or more domains.

Import all Parts

Imports all the domains as Parts.

Import selected Parts

Imports selected domains as Parts. Select the domains you want to import in the top box.

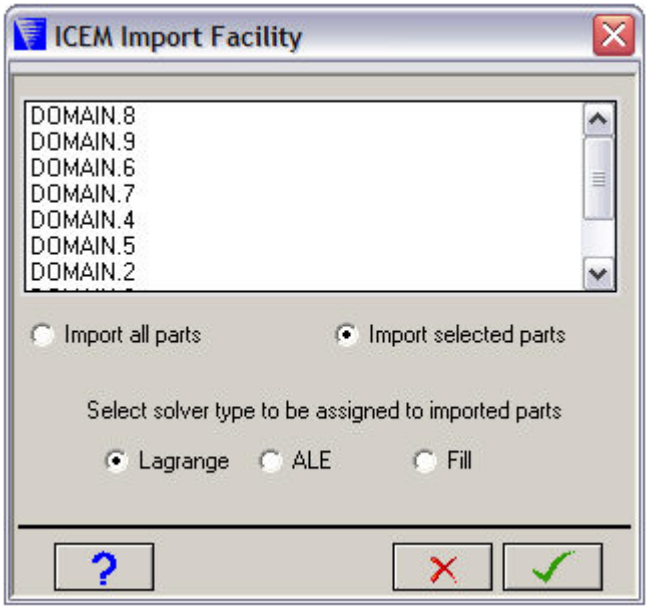
Select solver type

Select the solver type you want to assign to the imported Parts.

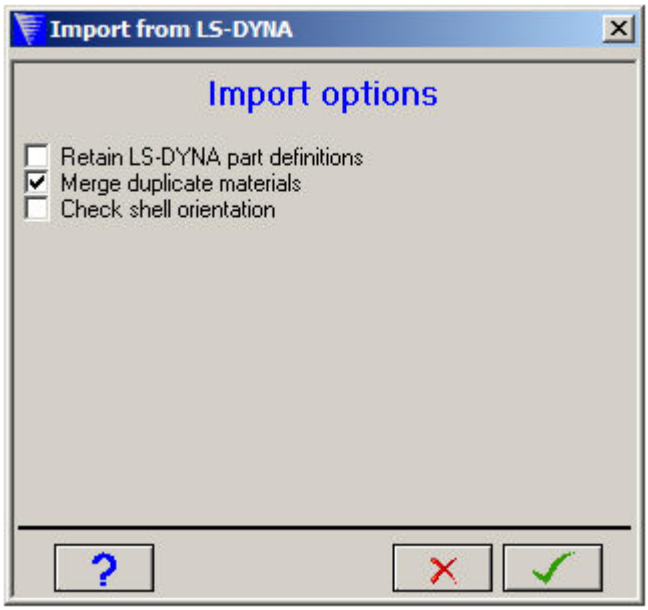
Click to import the Parts.

Parts are imported with their domain name. You can change these domain names later in the Parts panel.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



2.2.3. Pull-Down Menu - Import - LS-DYNA



Import Options

After browsing for an LS-DYNA keyword file, select import options in this dialog.

Retain LS-DYNA part definitions

Names the important parts from definitions within the LS-DYNA file. Otherwise, generic names are automatically generated for the parts.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Merge duplicate materials

Merges duplicate materials

Check shell orientation

Checks for inconsistent or non-manifold shell meshes. If the mesh is inconsistent and manifold, you should re-orient the mesh to have consistent normals.

LS-DYNA Keyword File Format

Both formatted and free formatted (comma delimited) LS-DYNA keyword files can be imported into Autodyn. The standard used is based on version 970 of LS-DYNA.

The primary use of the LS-DYNA reader is to allow you to define a mesh to be used with the unstructured solvers. In addition, other information regarding material definitions, boundary conditions and initial conditions can be imported and stored in Autodyn.

An overview of the information that can be imported into Autodyn from an LS-DYNA keyword file is given below.

Mesh

The mesh, represented as nodal positions and element connectivity, is read into Autodyn and sorted to produce Parts and Components. A Part is a collection of connected elements of the same classification (for example, volume elements, shell elements and beam elements). A Component is a collection of all connected elements regardless of their topology. By default, the defined Parts in Autodyn will not necessarily be equivalent to those defined in the LS-DYNA keyword file.

An option to retain the Part definitions in the LS-DYNA file is also available.

Materials

Each element defined in the keyword file will have an associated material definition. Corresponding materials will be automatically generated during the import of the keyword file into Autodyn. In general (except for the materials identified below), the actual material parameters will not be read into Autodyn and you should ensure that all materials are properly defined after the import.

At present, a subset of the material definitions and parameters are automatically read into Autodyn. These include materials of type rigid, elastic, piecewise linear plasticity and spring elastic material models.

Initial and Boundary Conditions

Fixed and prescribed boundary conditions are imported from the LS-DYNA keyword file into Autodyn. These are converted into the equivalent Autodyn boundary conditions including parameters, where possible.

Initial velocities defined on nodes and sets of nodes in the keyword file are read into Autodyn and converted into Initial Conditions. The initial condition is also automatically applied to the identified nodes.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Shell and Beam Sections

Shell sections defined in the keyword file are automatically transformed into Autodyn initial conditions. The local direction for layered shells is not currently translated.

Beam cross-sections and directions are read from the keyword file and these and the relevant data are converted to be compatible with beam types used in Autodyn.

Keyword Compatibility

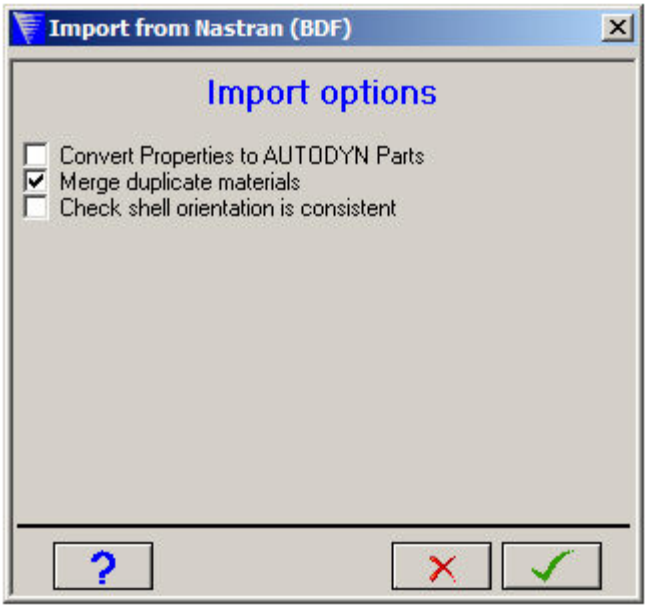
The current capabilities of the Autodyn import capability for LS-DYNA (.k) keyword files is summarized below:

Keyword	Compatibility
*BOUNDARY_SPC_NODE	Translational velocities only. DORFX, DORFY, DORFZ not recognized
*BOUNDARY_SPC_SET	Translational velocities only. DORFX, DORFY, DORFZ not recognized
*DEFINE_CURVE	Full
*ELEMENT_SOLID	Full
*ELEMENT_SHELL	Full
*ELEMENT_BEAM	Full
*INITIAL_VELOCITY	BOXID and IRIGID are not recognized
*INITIAL_VELOCITY_NODE	Translational velocities only. VXR, VYR and VZR are not recognized.
*INITIAL_VELOCITY_GENERATION	Translational velocities only. OMEGA, XC, YC, ZC, NX, NY, NZ and PHASE not recognized.
*MAT_PIECEWISE_LINEAR _PLASTICITY	Full
*MAT_ELASTIC	Full
*MAT_RIGID	Full
*MAT_SPRING_ELASTIC	Full
*NODE	Full
*PART	Full
*SECTION_BEAM	Beam types 1-5 supported. QR/IRID, SCOOR, NSM, NSLOC, NTLOC and IRR are not recognized.
*SECTION_SHELL	Shell thickness at first node is read in and set a shell thickness in initial condition.
*SET_NODE_LIST	Full
*SET_PART_LIST	Full
*SET_SHELL_LIST	Full

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Keyword	Compatibility
*SET_BEAM	Full
*SET_DISCRETE	Full
*SET_SOLID	Full
*SET_TSHELL	Full
*SET_NODE_LIST_GENERATE	Full
*SET_NODE_PART_GENERATE	Full
*SET_NODE_SHELL_GENERATE	Full
*SET_BEAM_GENERATE	Full
*SET_DISCRETE_GENERATE	Full
*SET_SOLID_GENERATE	Full
*SET_TSHELL_GENERATE	Full

2.2.4. Pull-Down Menu - Import - Nastran



Import Options

After browsing a NASTRAN input file, select import options in this dialog.

Convert Properties to Autodyn parts

Names the imported parts from definitions within the NASTRAN file. Otherwise, generic names are automatically generated for the parts.

Merge duplicate materials

Merges duplicate materials.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Check shell orientation is consistent

Checks for inconsistent or non-manifold shell meshes. If the mesh is inconsistent and manifold, you should re-orient the mesh to have consistent normals.

Input File Format

Autodyn can import MSC.NASTRAN™ input files in both free and fixed format. In free format, the entries can appear anywhere on the line and are separated by commas. With fixed format, a field must be located in a set part of the line. The fixed format field can be small (8 characters) or large (16 characters).

In addition to the MSC.NASTRAN™ bulk data input, Autodyn can recognize the expanded input data set for explicit transient dynamics, as given by MSC.DYTRAN™ version 4.7.

The primary use of the MSC.NASTRAN™ reader is to allow you to define a mesh to be used with the unstructured solvers. In addition, other information regarding material definitions, boundary conditions and initial conditions can be imported and stored in Autodyn.

An overview of the information, which can be imported into Autodyn from an MSC.NASTRAN™ bulk data file is given below.

Mesh

The mesh, represented as nodal positions and element connectivity, is read into Autodyn and sorted to produce Parts and Components. A part is a collection of connected elements of the same classification e.g. volume elements, shell elements and beam elements. A component is a collection of all connected elements regardless of their topology.

Materials

Each element, imported from the bulk data file will have the material name as defined on the property field for the element (PSOLID, PSHELL). Corresponding materials will be automatically generated during the import of the keyword file into Autodyn. In general (except for the materials identified below), the actual material parameters will not be read into Autodyn and you should ensure that all materials are properly defined after the import.

At present, a subset of the material definitions and parameters are automatically read into Autodyn. These include materials of type rigid, elastic, piecewise linear plasticity and spring elastic material models.

Initial and Boundary Conditions

Every element property is translated into an Initial Condition in Autodyn and allocated to the appropriate elements.

Fixed and prescribed boundary conditions are imported from the bulk data file into Autodyn. These are converted into the equivalent Autodyn boundary conditions including parameters, where possible.

Initial velocities defined on nodes and sets of nodes in the bulk data file are read into Autodyn and converted into Initial Conditions. The initial condition is also automatically applied to the identified nodes.

Bulk Data Compatibility

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

The current capabilities of the Autodyn import capability for MSC.NASTRAN™ bulk data files is summarized below:

Keyword	Compatibility
BEGIN BULKDATA	Full
CBAR	Only grid numbers to define local x-y plane
CBEAM	Only grid numbers to define local x-y plane
CHEXA	Full
CPENTA	Full
CQUAD4	No material direction
CTETRA	Full
CTRIA3	No material direction
JOIN	Full
FORCE	Full
GRID	No coordinate system; no SPC
PBAR	Full
PBEAM	Only constant cross-section
PBEAM1	Only as PBEAM
PCOMP	Full
PLOAD	Full
PSHELL	No thickness
PSHELL1	Up to field 7
PSOLID	Full
SET1	Full 1
SPC	Full
SPC1	No ranges
TABLED1	Full
TLOAD1	Only FORCE

Any keyword not mentioned in the table above will be ignored.

2.2.5. Pull-Down Menu - Import - Convert IJK to Unstructured

Existing structured (IJK) Parts using the Lagrange, Shell and Beam solvers can be converted into Unstructured Parts via the Import, Convert IJK Part to Unstructured option. The mesh, material, boundary conditions and initial conditions will be transferred to the new Part.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

This feature lets you convert existing models to the new solvers to take advantage of the reduced memory requirements and increased speed of solution.

Note:

This option is only available for 3D analyses.

During the conversion, you will be presented with the options:

Delete converted IJK regions

Deletes the structured IJK Parts that have been converted to unstructured solvers after the conversion process.

Merge connected IJK Parts into single Unstructured Part

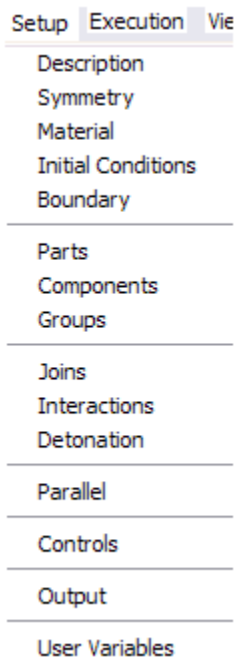
Complex geometries defined using the structured (IJK) solvers tend to be made up of a large number of relatively simple Parts that are joined together (for example a mesh imported from ICEM-CFD `.geo` file). By default, the conversion process will merge connected IJK Parts into a single Part. This simplifies the model and can also improve performance because there will be fewer Joins in the model. For some applications, you may not wish to merge the IJK Parts in this way. Clearing this option will result in no change in the Parts defined in the model.

Merge joined IJK nodes into single unstructured node

By default, all joined nodes are merged into a single unstructured node during the conversion process. In some cases, for example if you have pinned joins, you may not wish to do this.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

2.3. Pull-Down Menu - Setup



Enables you to set up models. Most of these options are accessible through the Navigation Bar.

Description

Enables modification of the description of your model.

Symmetry

Change the symmetry settings for your problem.

Material

Define and modify materials.

Initial Conditions

Define and modify Initial Condition Sets

Boundary

Define and modify Boundary Conditions.

Parts

Define and modify Parts

Components

Define and modify Components

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Groups

Define and modify Groups

Joins

Define joins.

Interactions

Define interactions

Detonation

Define detonation points.

Parallel

Set up your parallel processing environment and prepare your model for parallel processing.

Controls

Set various controls for your model such as run limits, timestep controls, damping constants, cutoffs and gravitational forces.

Output

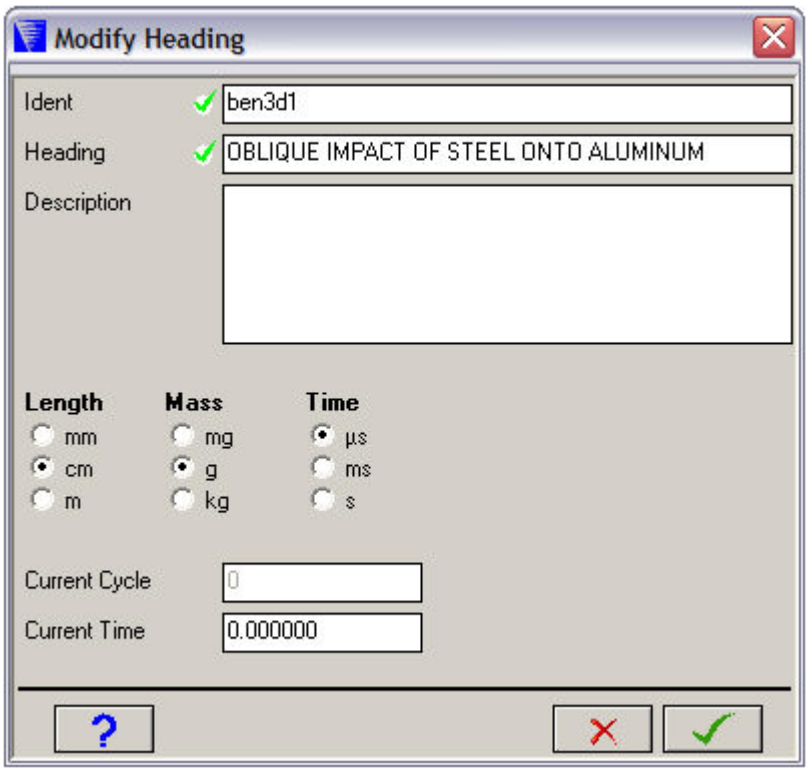
Select the output you want from your model.

User Variables

Initialize any grid variables you want to use in your user subroutines.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

2.3.1. Pull-Down Menu - Setup - Description



This window lets you modify the description of your model.

Ident

Changes the ident for your model.

The ident you choose is used to identify all the files created for your model.

For example, if your Ident is **MyImpactProblem**, then the cycles zero Save file for the problem will be named **MyImpactProblem_0.ad**.

Heading

Changes the heading for your model.

Headings appear in plots and other output for your model.

Description

Adds or changes a detailed description of your model.

Units

Changes the units of length, mass and time you want to use for your model.

The default units (mm, mg, ms) will give good results, with small rounding errors, for most models.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Changing the units here *will not* result in any scaling of data previously entered. It affects only the units displayed in output and data conversions performed when reading / writing data to/from material libraries.

Current Cycle

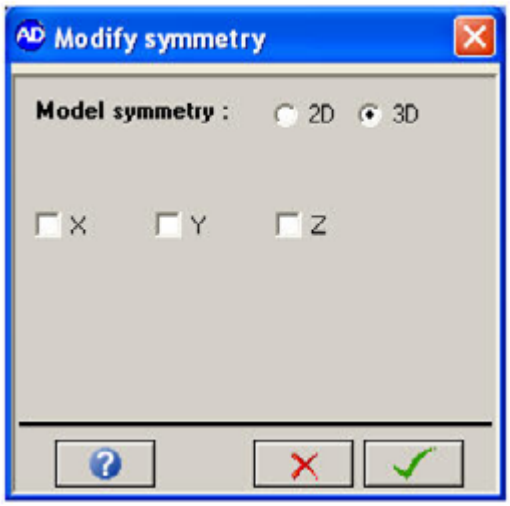
Displays the current cycle number.

Current Time

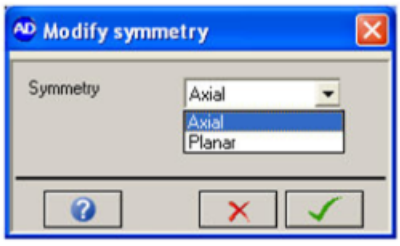
Changes the current time.

2.3.2. Pull-Down Menu - Setup - Symmetry

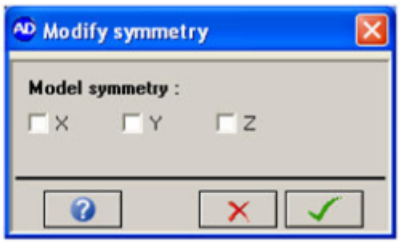
Autodyn started up in Workbench:



Autodyn started up from outside Ansys Workbench:



2D Window

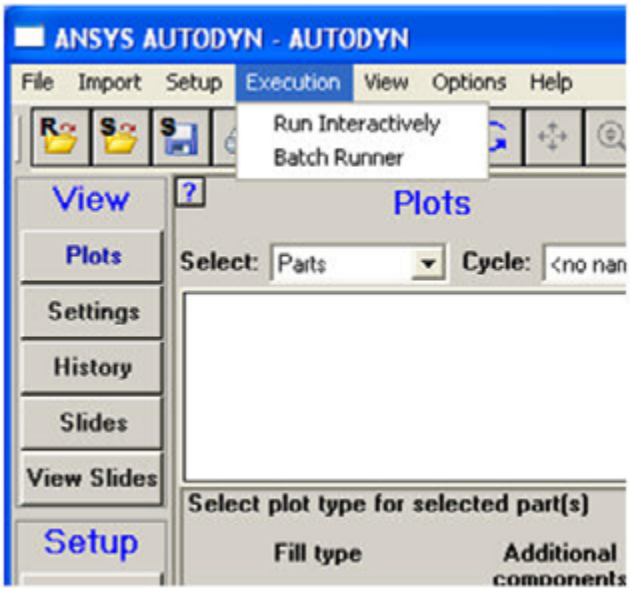


3D Window

This window lets you change the symmetry used in your model.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

2.4. Pull-Down Menu - Execution



Enables you to execute your calculation.

Run Interactively

Executes your problem interactively. This is the same as clicking on the Navigation bar.

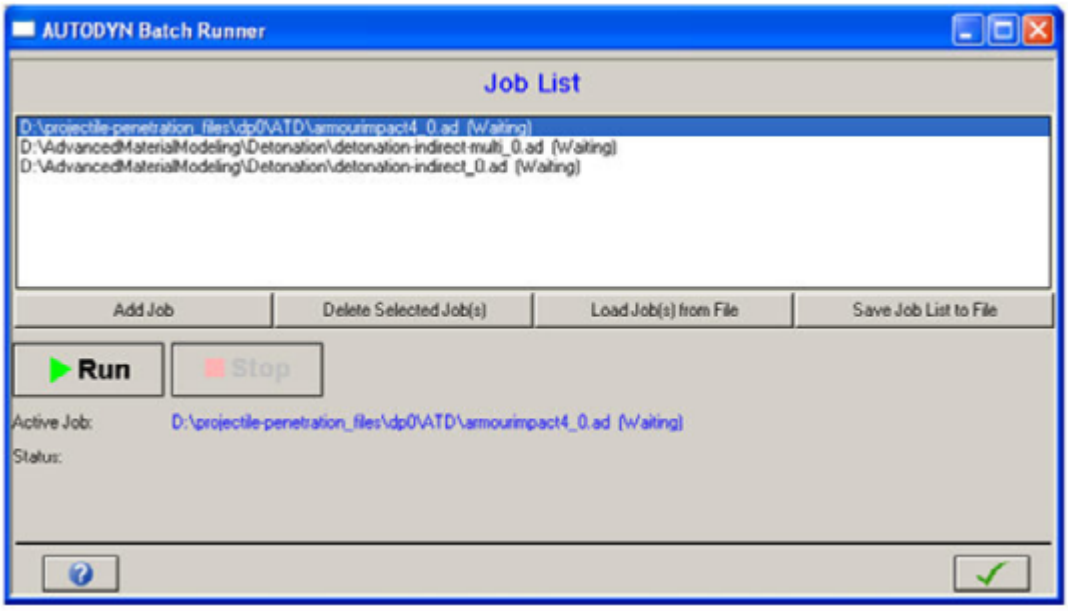
This is the same as clicking on the Navigation bar.

Batch Runner

Runs one or more of your models in batch mode (that is, without graphics).

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

2.4.1. Pull-Down Menu - Execution - Batch Runner



This window lets you set up and run single or multiple jobs (calculations) in batch mode (without graphics).

Job List

Lists all current jobs (that is, Save files that are to be loaded and executed) and shows their status.

Add Job

Adds a job to the list. A window opens to let you browse to the Save file you want to load and execute.

Delete Selected Job(s)

Select one or more Jobs in the Job List and click this button to delete them from the Job List.

Load Job(s) from File

Loads jobs saved in a file (`.bat`) into the Job List.

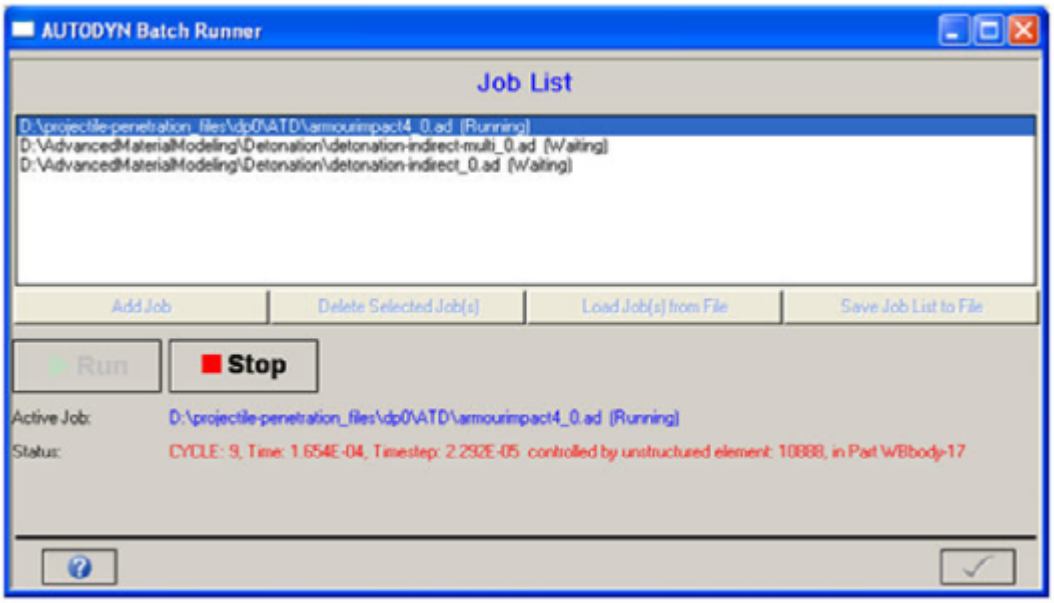
Save Job List to File

Saves all the jobs in the current Job List to a file (`.bat`).

Run

Runs the jobs in your Job List.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Jobs run sequentially in the Job List.

While running, the window displays information on which jobs have completed, which job is currently executing and which are waiting in line to execute. The status of the executing job is also displayed at the foot of the window. At the start of each job, the solver performs standard checks on model integrity. In interactive mode, you may be required to acknowledge or decline certain decisions, but when using the Batch Runner all decisions are automatically accepted. These decisions are recorded in the log file.

Stop

Stops running jobs in the Job List.

2.5. Pull-Down Menu - View

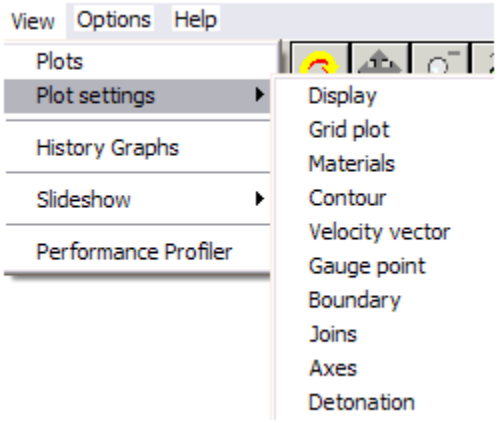
Enables you to customize your view (what you see in the View panel). Most of these options are accessible through the Navigation Bar.

Plots

Sets up what you want to see in the View panel.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Plot Settings



Lets you change settings for the various plot types that control how you view your model.

You can change settings for the following plot types:

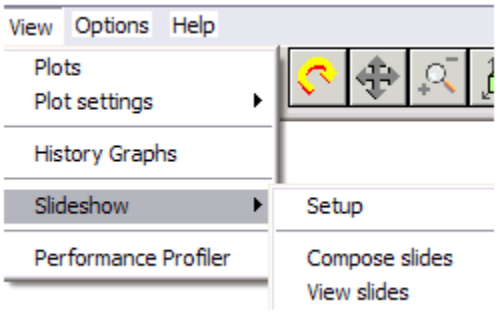
- **Display**
- **Grid**
- **Materials**
- **Contour**
- **Velocity Vector**
- **Gauge Point**
- **Boundary**
- **Joins**
- **Axes**
- **Detonation**

History Graphs

View histories of Gauge points and Material, Part and Energy summaries.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Slideshow



Creates and organizes slideshows (animations)

Performance Profiler

Displays a performance profiler that lets you monitor your calculation.

2.5.1. Pull-Down Menu - View - Slideshow - Setup

This window lets you set parameters for creating a slideshow.

Directory

Click next to the Directory box to browse to the directory in which you want to save your slideshow.

Slideshow prefix

Enter a slideshow prefix in this field or check **Use current model name** to use the current Ident as the slideshow prefix.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

This prefix is used for all image files created for the slideshow.

Next slide number

Enter a number for the next slide to be created or check **Automatic numbering** if you want Autodyn to automatically assign numbers to slides.

Image type

Use this pull-down menu to select the image type you want to use to create your slides (gif, jpeg, tiff, or gfa).

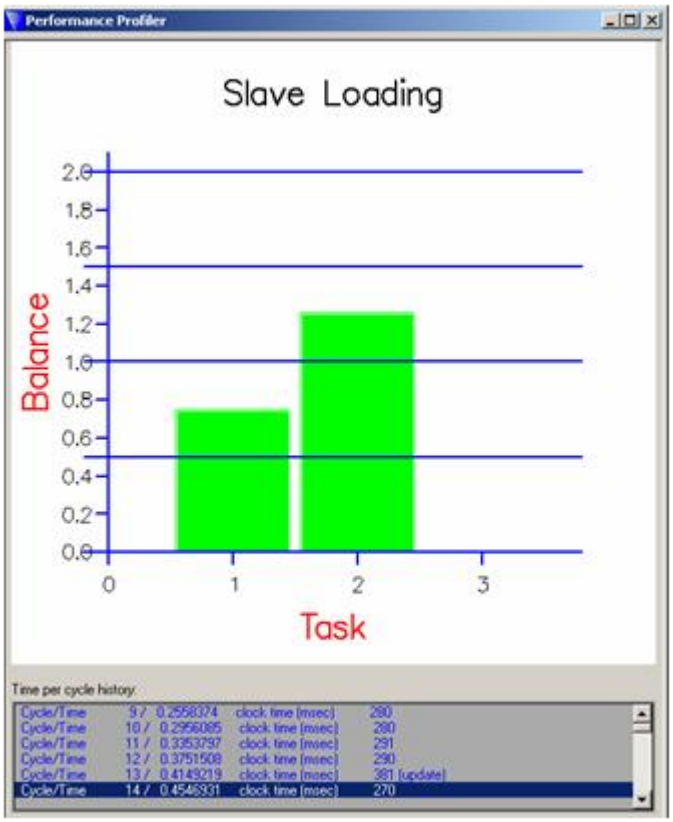
Use OpenGL

Check this box if you want to use OpenGL to create your slides. This can significantly improve the plotting of SPH objects.

Fix Slide size

Select this toggle to fix the size of slides created. Specify the width and height of the image in pixels. If the toggle is not selected, the slides created are to be the size of the graphics window.

2.5.2. Pull-Down Menu - View - Performance profiler



This window provides a Performance Profiler that lets you visually monitor the load balancing performance of parallel simulations.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

To take full advantage of this option, you must create an empty file **profile.cfg** in the directory where the Autodyn model resides.

If you do this, the following type of output will be displayed in the scrolling box at the foot of the window:

```
Cycle/Time    1  /  1.637809225157053E-4 clock time (msec) 32856  (update)
Cycle/Time    2  /  3.439399411878227E-4 clock time (msec) 17030
Cycle/Time    3  /  5.421148660224775E-4 clock time (msec) 17056
Cycle/Time    4  /  7.601072880654561E-4 clock time (msec) 17029
Cycle/Time    5  /  9.998989575100770E-4 clock time (msec) 17054
Cycle/Time    6  /  1.263669799616239E-3 clock time (msec) 17016
Cycle/Time    7  /  1.553817732221804E-3 clock time (msec) 17081
Cycle/Time    8  /  1.872980465005591E-3 clock time (msec) 16994
Cycle/Time    9  /  2.224059478677190E-3 clock time (msec) 17116
Cycle/Time   10  /  2.610246402086324E-3 clock time (msec) 17024
Cycle/Time   11  /  3.035052027043786E-3 clock time (msec) 17075
Cycle/Time   12  /  3.502338224625149E-3 clock time (msec) 17070
Cycle/Time   13  /  4.016353053105620E-3 clock time (msec) 17053
Cycle/Time   14  /  4.581769376689206E-3 clock time (msec) 17019
Cycle/Time   15  /  5.203727346111727E-3 clock time (msec) 17070
Cycle/Time   16  /  5.887881127305133E-3 clock time (msec) 17011
Cycle/Time   17  /  6.640450302929377E-3 clock time (msec) 17085
Cycle/Time   18  /  7.468276414058693E-3 clock time (msec) 34853  (update)
Cycle/Time   19  /  8.378885156037852E-3 clock time (msec) 17017
Cycle/Time   20  /  9.380554793925532E-3 clock time (msec) 17098
```

Each line gives the cycle number, the cycle time, and the wall clock time that the calculation used to compute the cycle (in milliseconds). If the line ends with "**(update)**", this indicates that work unit information for the interaction logic was updated on that cycle. Such cycles usually take longer to compute.

The profiler also shows you visually how well balanced your calculation is. In the example shown above, **Balance** is a measure of the relative CPU time spent on each task for each cycle in the simulation. So a balance of 1.0 for each task is the ideal / most efficient goal. A balance of less than 1.0 means that a task is under utilized (does not have enough work to do). A balance of greater than 1.0 means that a task is over utilized (has too much work to do).

2.6. Pull-Down Menu - Options

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Enables you to set various GUI options.

Interactivity

Sets how the mouse and keyboard is used for the toolbar operations that manipulate images in the view panel.

Clipping Planes

Sets clipping planes for your plot.

Default Settings

Change the default settings used to start up Autodyn.

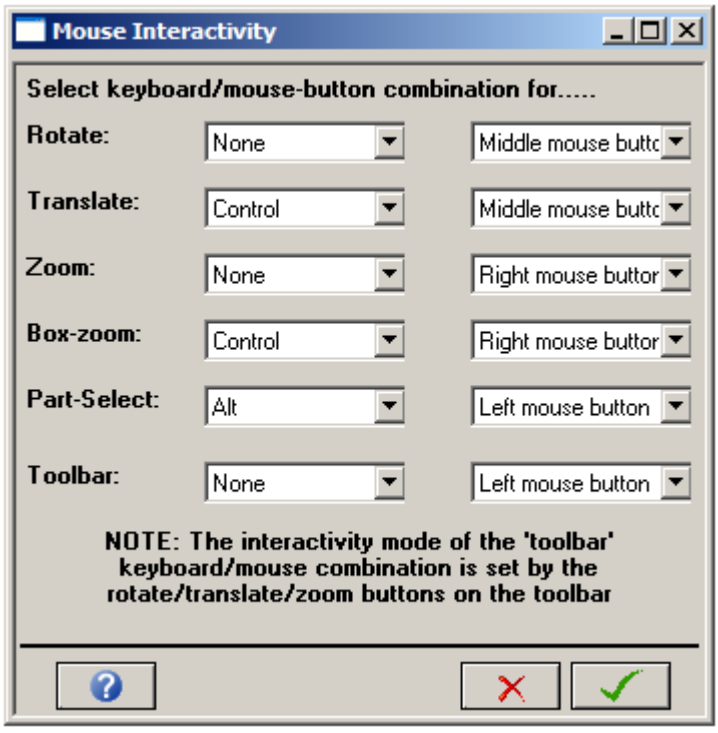
Pause License

Temporarily pauses the license and frees it for another Autodyn instance.

Create User-Subroutine Project

Installs the Autodyn project to create your own version of Autodyn by using user-subroutines.

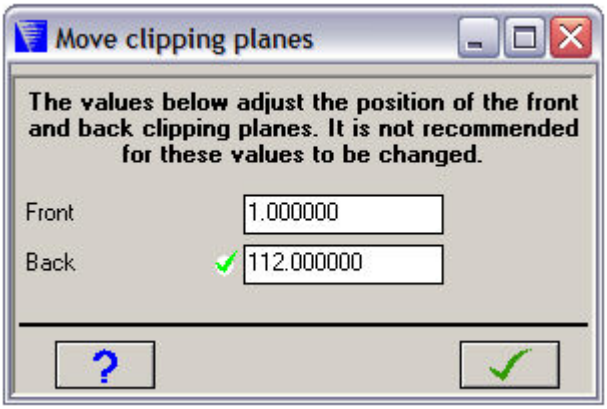
2.6.1. Pull-Down Menu - Options - Interactivity



This window lets you change how the mouse and keyboard is used for the toolbar operations that manipulate images in the view panel.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

2.6.2. Pull-Down Menu - Options - Clipping Planes



This window lets you set clipping planes for your plot.

If you zoom in close to, or into a grid, the image will not look good if you include everything in your view. The default values for the clipping planes usually improve such plots considerably, but there may be times when you want to use other values.

Front

Use this field to set the distance of the front clipping plane from the viewpoint. Any part of the grid nearer to the viewpoint than this plane will not be plotted.

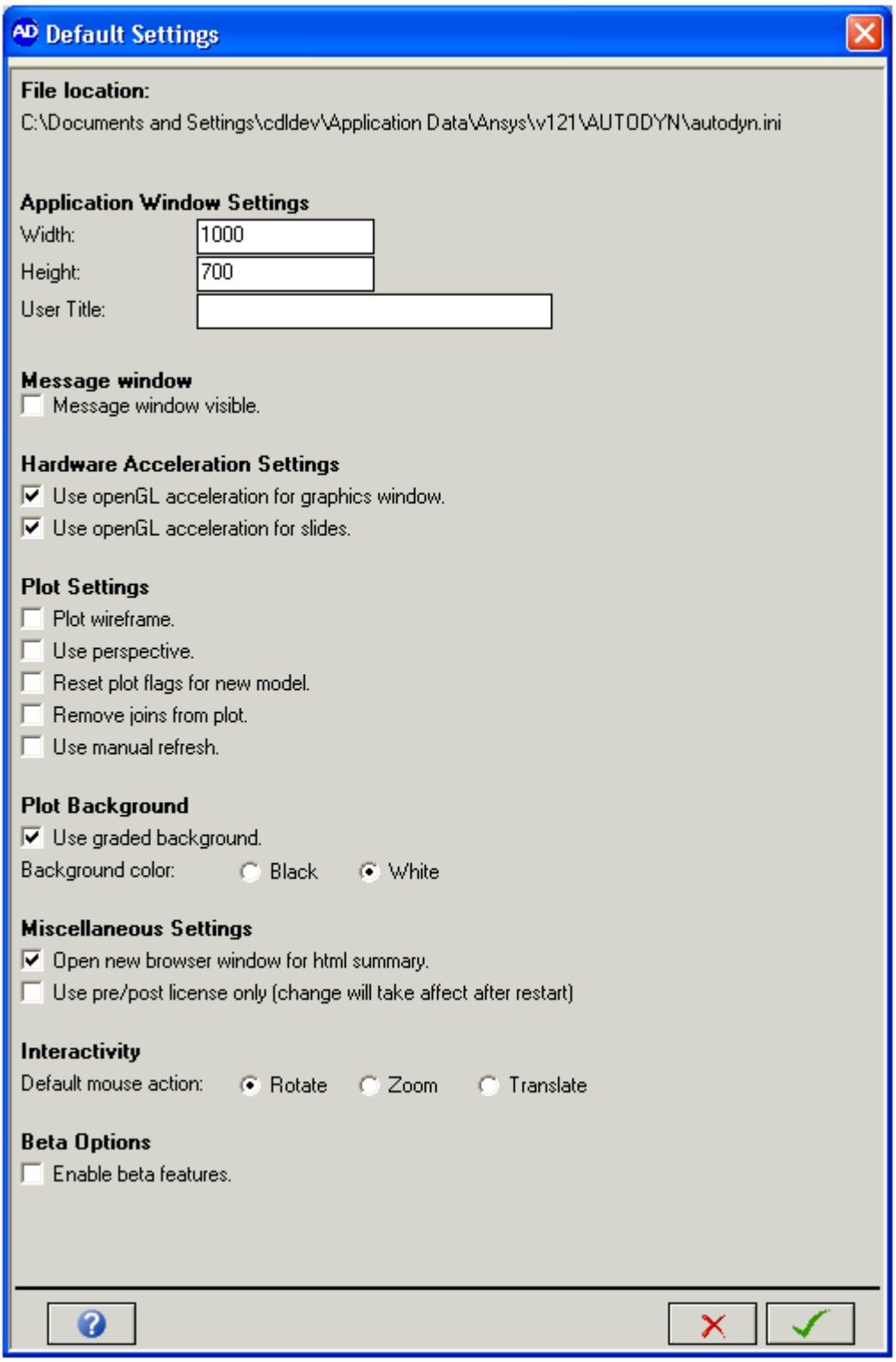
Back

Use this field to set the distance of the back clipping plane from the viewpoint. Any part of the grid further away from the viewpoint than this plane will not be plotted.

2.6.3. Pull-Down Menu - Options - Default Settings

This dialog lets you to define the default settings for the Autodyn GUI. These settings are stored in an ASCII file "autodyn.ini" and are used to initialize various aspects of the GUI and rendering options, each time you start Autodyn.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



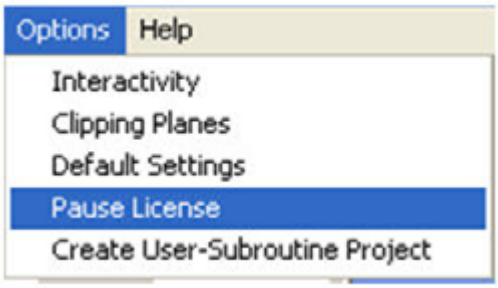
2.6.4. Pull-Down Menu - Options - Pause License

If you are working with a project in Workbench which contains an Autodyn system linked to an Ansys Explicit Dynamics system you will not be able to open and work in both applications simultaneously in shared licensing mode or when the license server has only one solver license available. For example,

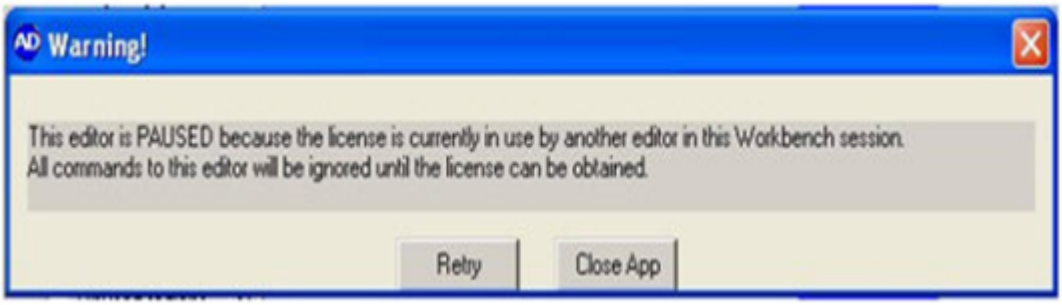
Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

if you want to generate a new mesh in the meshing application then you will need to close the Autodyn application to release the license for the meshing application to use.

As an alternative, you can leave the Autodyn application open, but release the license for another application to use. While the license is released, the Autodyn user interface will be locked.



After selection of the Pause option the license is released, and the Autodyn user interface will be locked. To continue the Autodyn application, you can select Retry to try to obtain the license again. The Close App option will exit the Autodyn application without saving any data.



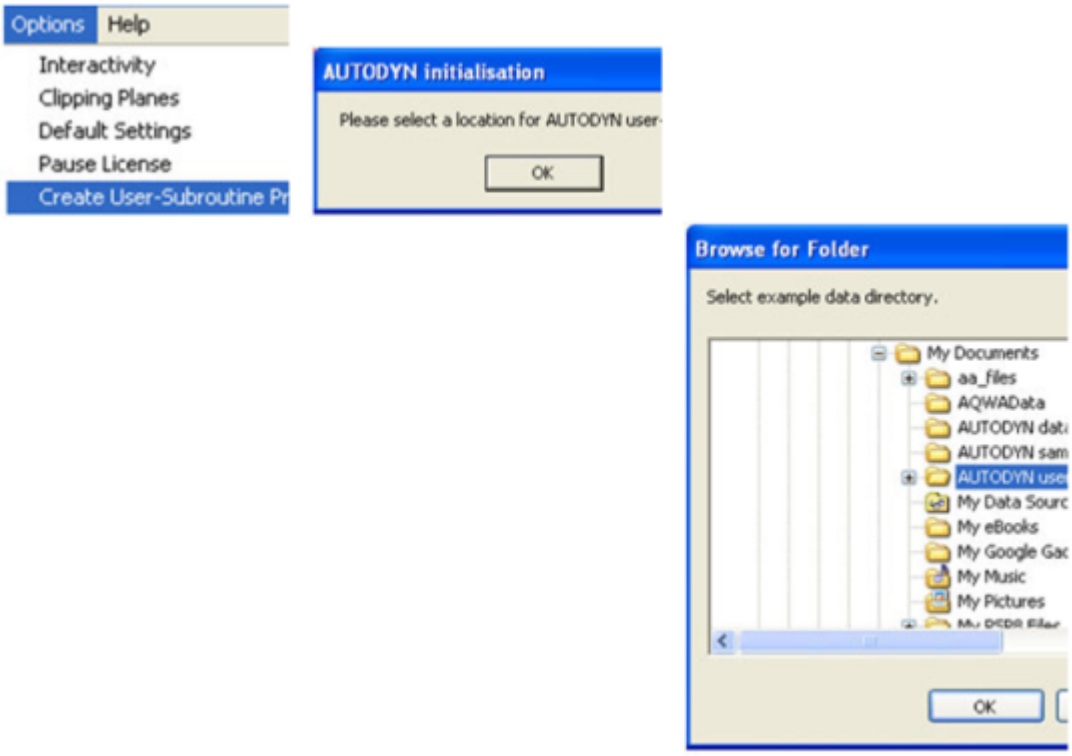
For more information on licensing, review the *Ansys Licensing Guide*.

2.6.5. Pull-Down Menu - Options - Create User-Subroutine Project

When Autodyn is run for the first time by the logged on user, Autodyn will perform a few additional initialization steps to create user-specific settings.

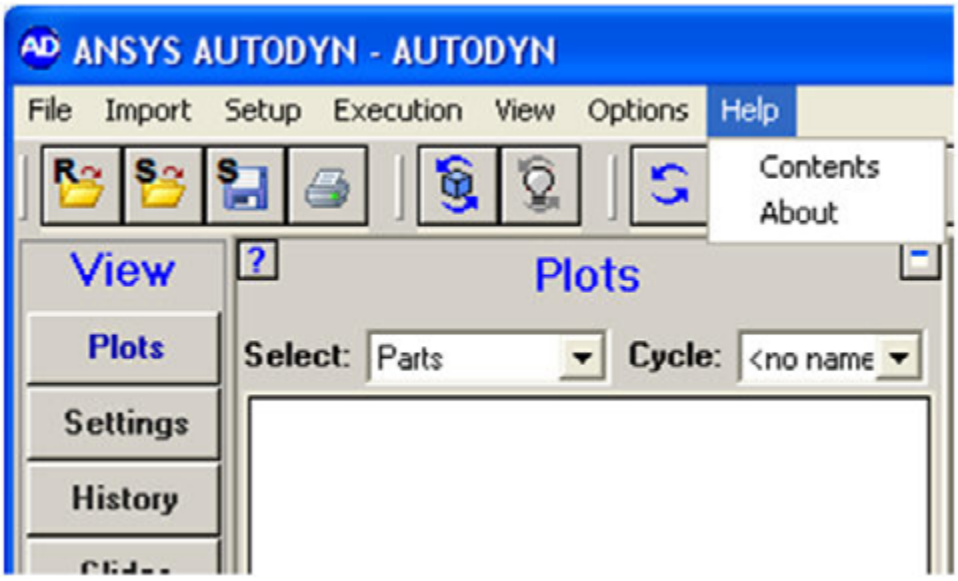
In previous version you also would have been prompted for a location for user-subroutine files which would have been created after the initial step. This step is no longer performed and the generation of the user-subroutine project is now available as a user action from within Autodyn from the Options Menu instead.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



After selection of the location for the user-subroutines, Autodyn will install the user-subroutine project in a subdirectory with name Autodyn *user_subroutines*.

2.7. Pull-Down Menu - Help

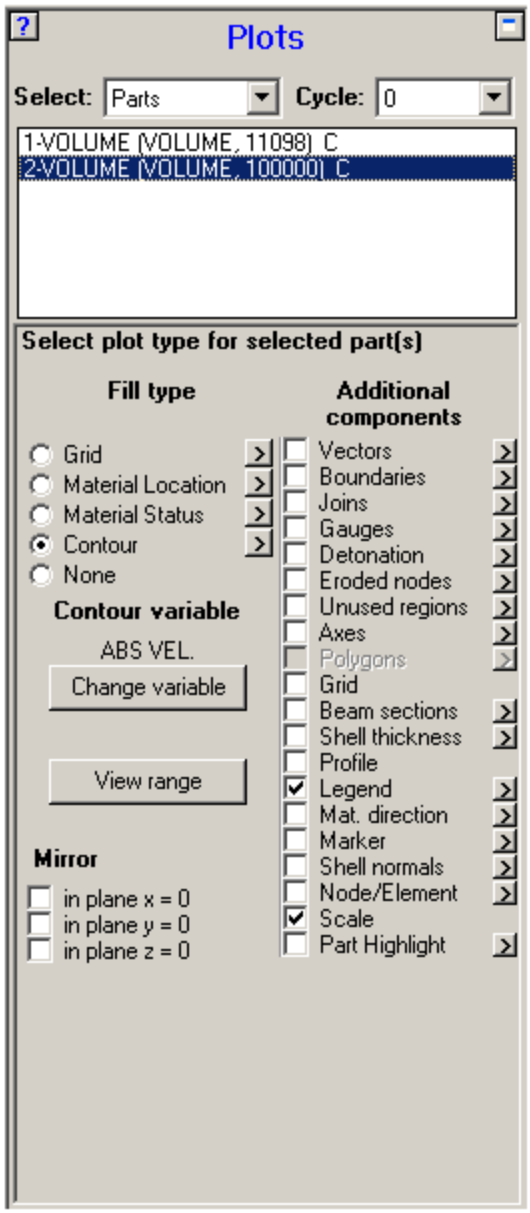


This pull-down menu provides general help for Autodyn (Contents), and provides information about the current version (About).

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information
of ANSYS, Inc. and its subsidiaries and affiliates.

Chapter 3: Autodyn Plots



This panel lets you set up what you want to see in the View panel.

Cycle

Choose which cycle you want to view for the current model (Ident).

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Select part(s)

The Parts in the model. Operations performed in the Plots panel will be applied only to the Part(s) you select.

Fill type

The basic fill type you want to view. Select only one fill type.

Additional components

View additional components in your plot by checking the boxes alongside the additional components you want to include (any number can be selected).

Contour Variable

Active only if a new Contour Fill type is selected. Click **Change Variable** to change the variable to be contoured.

View range

Active only if a single Part is selected. Click this button to limit the IJK range that is plotted for the Part.

Mirror

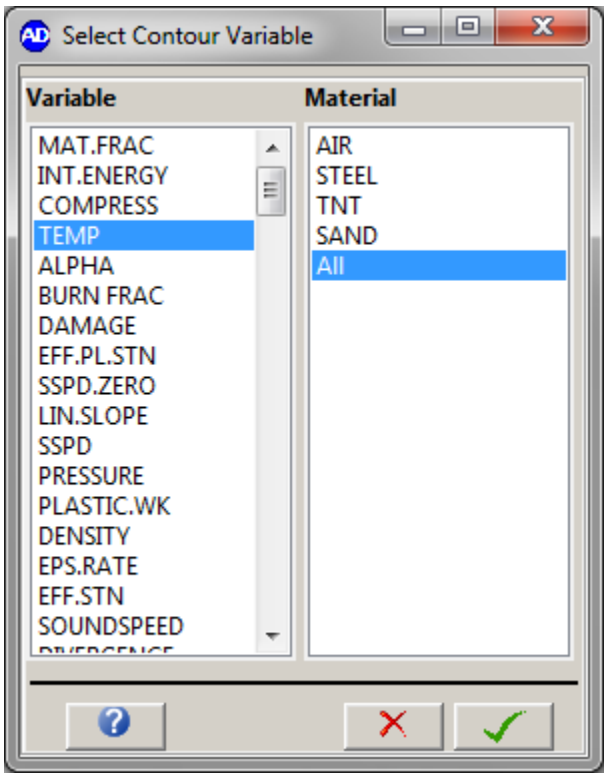
Check a box to mirror the image in the associated axis.

All Fill types and Additional components have default **Settings**. You can quickly access and change these settings by pressing the button to the right of these options. Doing so will bring up a dialog window containing the settings available for the associated option.

The **Settings** button on the Navigation bar gives you access to all these settings.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

3.1. Plots - Select Contour Variable



This dialog lets you select the variable you want to contour.

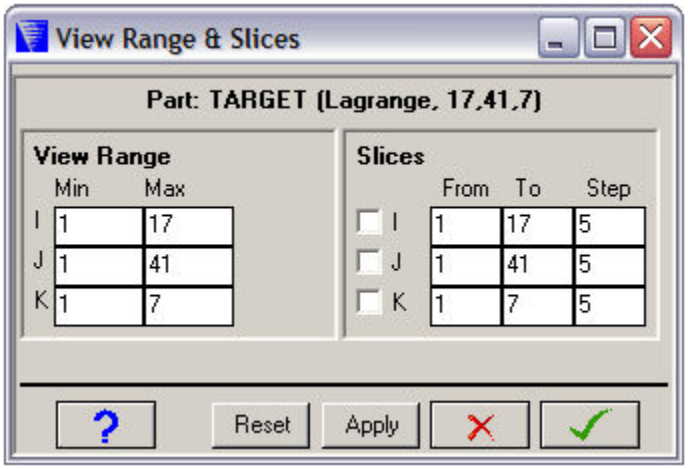
Select the variable from the left hand list.

For multi-material variables (those at the top of the left hand list above the variable PRESSURE), you must also select a material (or **All**) from the right hand list. When a material is selected, the value for that material fraction in an element is plotted. When All is selected, a mass-weighted average for all material fractions in an element is plotted.

For NBS tetrahedral elements, contour and element gauge plots of standard variables are a volume weighted average of the values at the nodes of that element. An additional variable set, with suffix '.NBS', can be used to plot the SCP solution used to stabilize the nodal solution for parts with a non-zero Puso stability coefficient. This variable set can also be used to plot variable values (average over all materials) at node gauges.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

3.2. Plots - View range and slices (Structured)



This dialog window lets you set the view range and slices to be used to display a structured part of your model.

Part

The part for which you are defining a view range and slices is shown here, together with its index range.

View Range

Define the I, J, and K range you want to see for this Part.

Slices

Instead of a solid view of your Part, you can request to see only slices (planes) through the part for any of the three indices.

Check the box next to an index for which you want to create slices and define the planes for which you want to see these slices.

If no boxes are checked, a solid plot using the view range will be shown.

Reset

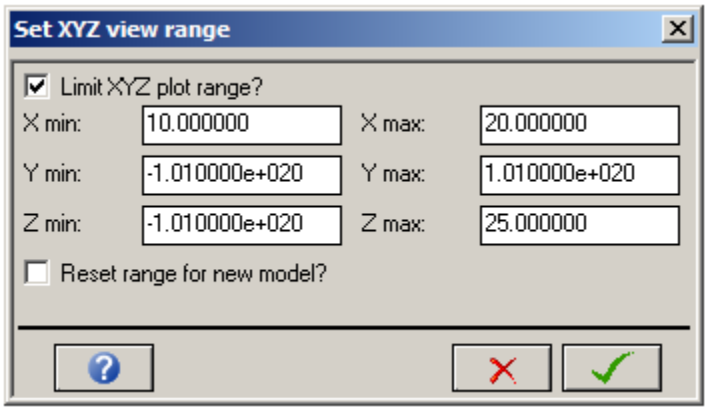
Resets the View Range and Slices to their default values.

Apply

Applies the current settings.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

3.3. Plots - View range and slices (SPH/Unstructured)



Limit XYZ plot range?

Truncates plotting of unstructured and SPH parts. Any unstructured elements or SPH nodes that fall outside the view range specified will be eliminated from the plot.

View Range

Specify the maximum and minimum X, Y, and Z coordinates to be plotted for unstructured and SPH parts. These limits are not set on a per part basis, and so setting limits will truncate the plotting for all unstructured and SPH parts in the model.

Reset range for new model?

If this option is selected, when a new model is opened, the view range will be reset to show the full model. If this option is not selected, the view range you have selected will be maintained.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

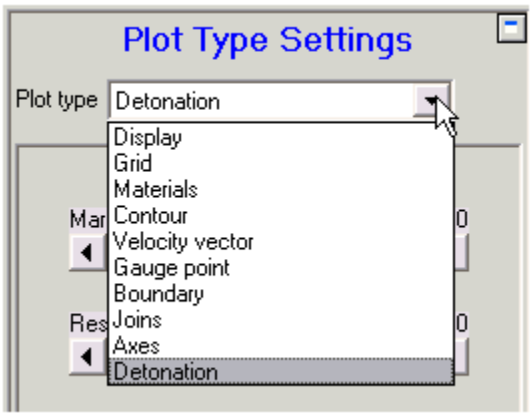
Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information
of ANSYS, Inc. and its subsidiaries and affiliates.

Chapter 4: Autodyn Settings

Selecting **Settings** on the Navigation Bar displays the **Plot Type Settings** panel.

This panel lets you change settings for the various plot types that control how you view your model.

At the top of this panel, a pull-down menu lets you to select the plot type whose settings you wish to change :



You can change settings for the following plot types:

- **Display**
- **Grid**
- **Materials**
- **Contour**
- **Velocity Vector**
- **Gauge Point**
- **Boundary**
- **Joins**
- **Axes**
- **Detonation**

These settings can also be changed by pressing next to the option in the **Plots** panel.

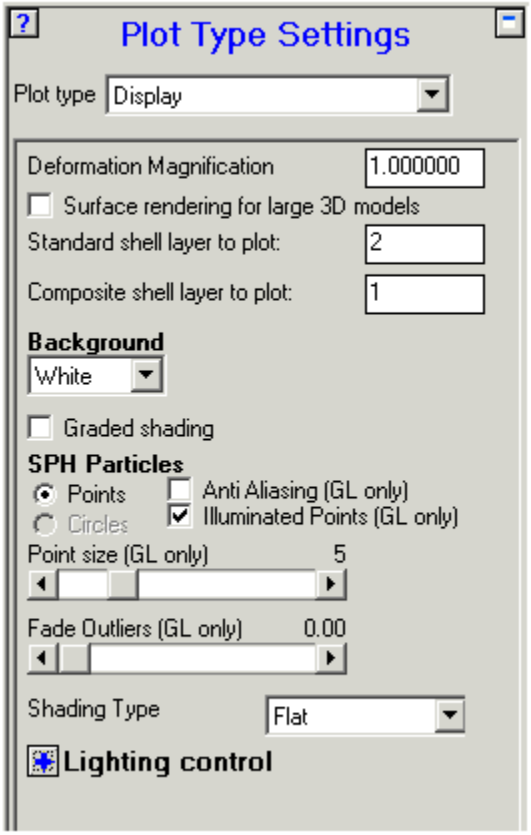
Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

The following additional settings can also be changed by pressing next to the option under **Additional Components** in the **Plots** panel.

- **Eroded Nodes**
- **Unused Regions**
- **Polygons (2D only)**
- **Beam sections**
- **Shell thickness**
- **Legend**
- **Profile**
- **Mat. direction**
- **Marker**
- **Shell normals**
- **Node/Element**
- **Part Highlight**

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

4.1. Settings - Display



This panel lets you set the following general options :

Deformation Magnification

View magnified grid deformations. Input the magnification you want to use (1.0 gives no magnification).

Surface rendering of large 3D models

3D models using hexahedral elements (e.g. Lagrange or Euler) are usually rendered using all the elements. Check this box to render models using only external faces. This makes the process much quicker. This option cannot be used for all plots (e.g. plots with opacity applied).

Standard shell layer to plot

Set the layer that is plotted for standard shells in material location plots, material status plots, and material contour plots.

Composite shell layer to plot

Set the layer that is plotted for composite shells in material location plots, material status plots, and material contour plots.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Background

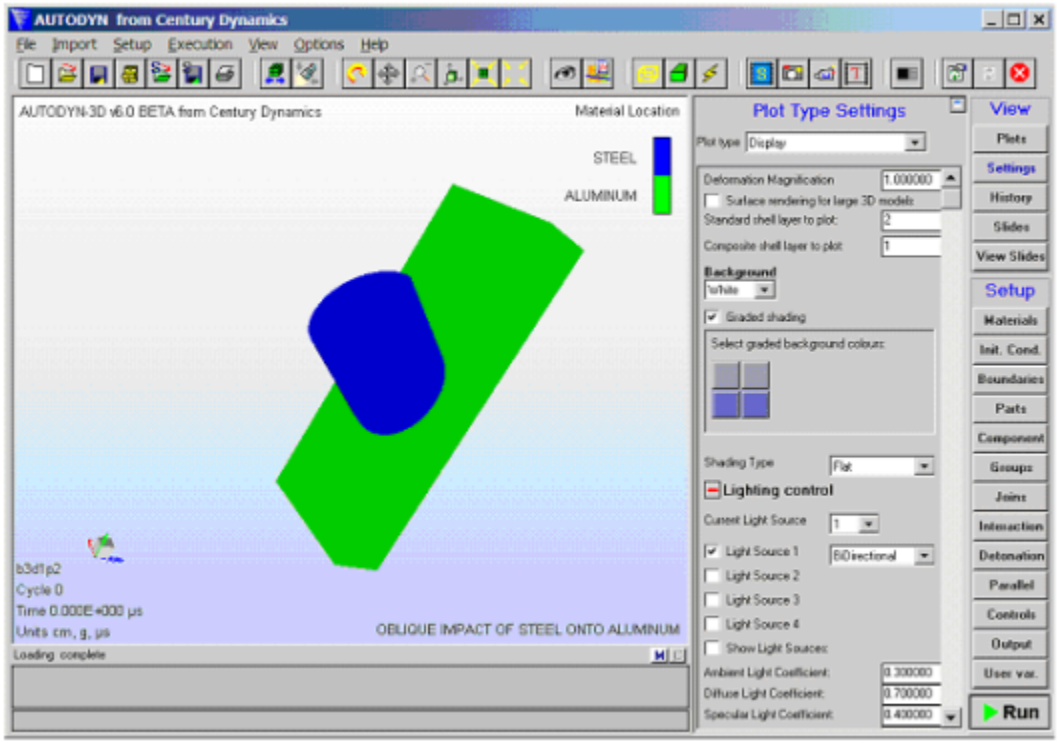
Set the background color for the view panel.

Graded Shading

Control the color, including color grading, for the background of the View panel.

You may select a color to be used for each of the four corners of the display by pressing the buttons:

The selected (Default) colors shown here will produce the following background:

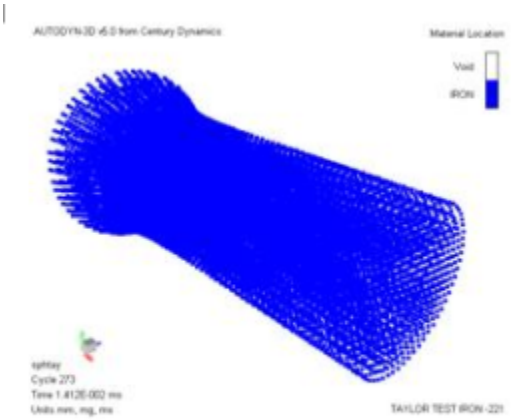


SPH Particles

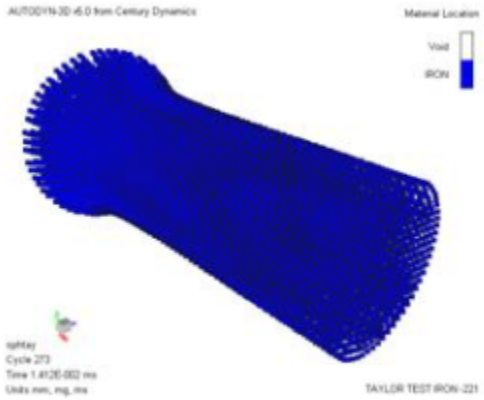
Controls for the display of SPH particles are only shown when the model contains SPH parts. In this section, the controls for displaying the points with anti-aliasing and illuminated points are accessible. In addition, control of particle display size and amount of fade for outliers can be set.

The figures below compare plots with and without the illuminated points option active:

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



SPH representation without Illuminated Points



SPH representation with Illuminated Points

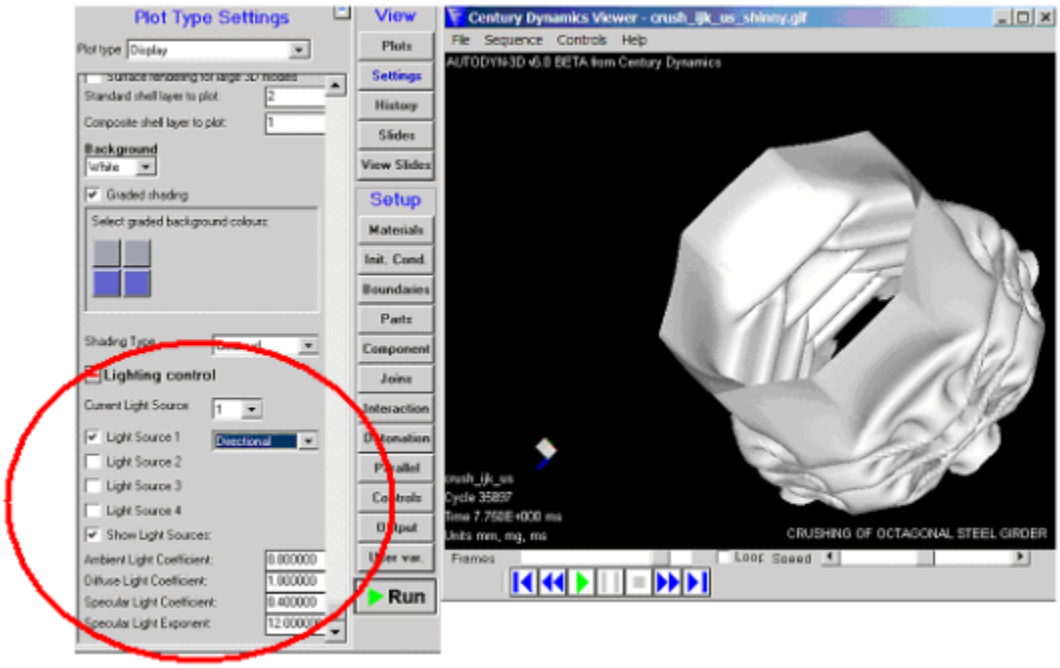
The fade outliers slider gives you the ability to fade in and out of view of the SPH particles, which exist outside of the range that is set in the contour panel when using the truncate facility. This facility is only available when illuminated points are displayed.

Note:

Unchecking the **Illuminated Points** option can help resolve the issue concerning the disappearance of the Ansys logo for models with SPH bodies in Autodyn. Once the **Illuminated Points** option is unchecked, you may need to click another button in the Navigation bar (for example, Plots) for the logo to reappear.

Advanced Image Shading and Lighting Options

A set of advanced shading and lighting options is available



Using a combination of the above options, metallic looking surfaces can be rendered, for example.

Shading type

The shading type can now be selected as either flat or Gouraud. For flat shading every point on a face is given the same intensity. For Gouraud shading, the mesh is shaded so as to give the appearance of curved surfaces. The intensity for each point is interpolated from the normal of the face vertices.

Flat shading

Gouraud shading

Lighting Control

The advanced lighting control lets you specify up to 4 different light sources and also to control the direction and form of each light source.

The following options are available:

- **Current light source**

  Select the current light source to be interactively transformed, after selecting the "Transform Light" button on the toolbar.

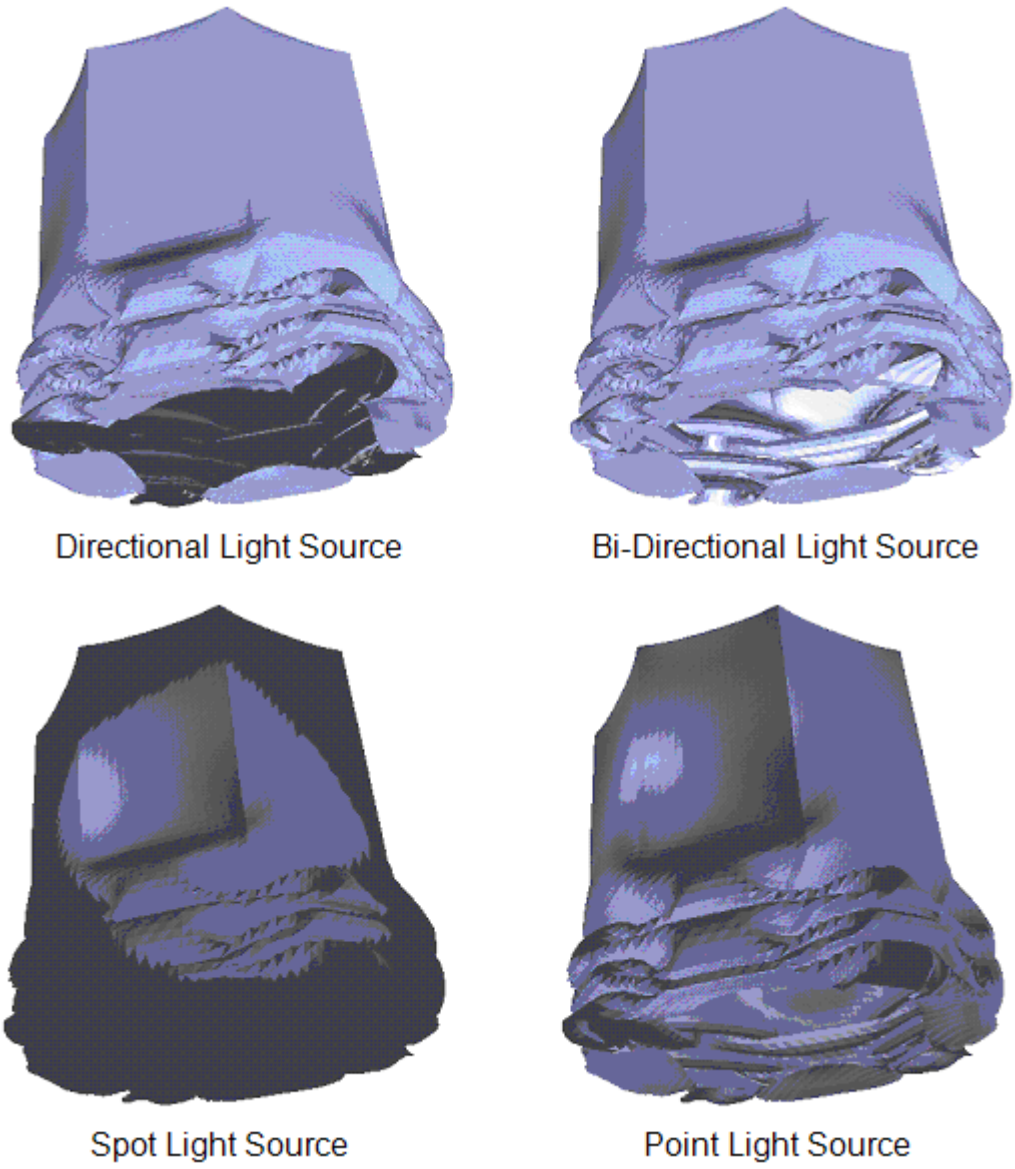
- **Light source toggles**

  The number of light sources required (up to a maximum of 4).

- **Light source type**

  Select between directional, bi-directional, spot, or point light sources. Examples of these are shown in the figure below.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



- **Show light sources**

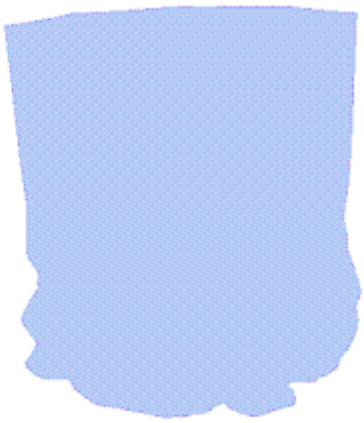
  The direction and position of the light sources is shown on screen by white arrows (switch the background color to black). Examples are shown in the figure below (from left to right) for bi-directional light source, spot light source, point light source and directional light.

- **Ambient, Diffusive and Specular Light Coefficients**

  The ambient light coefficient controls the amount of background lighting. Ambient light is non-directional, affecting all parts of all surfaces equally.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

No lighting



Ambient lighting only

Diffuse light reflectance controls the proportion of the available non-ambient light that the object reflects equally in all directions. Non-ambient light emanates from the directional and point light sources.

Specular highlights of a particular color and brightness are created when the direction of incoming light (from a directional or point light source) is "sufficiently close" to the viewing direction. The specular reflection coefficient determines the brightness of such highlights.

Diffusive light only

Specular light only

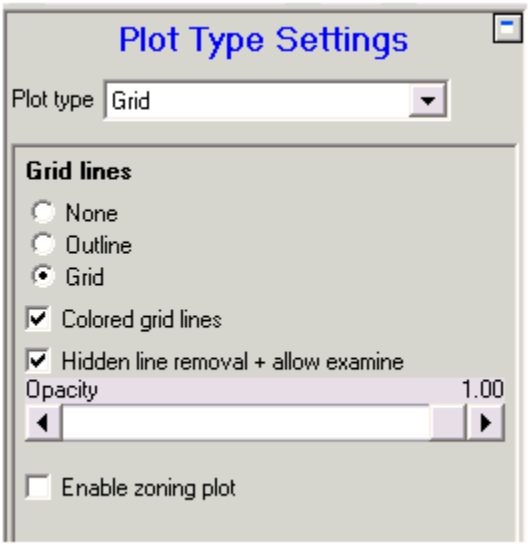
The specular exponent determines what "sufficiently close" means. The greater the sharpness, the smaller (more focused) the size of the specular highlight.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

High specular exponent

Low specular exponent

4.2. Settings - Grid



This panel lets you set the following options for grid plots (some options are not present for 2D models) :

Grid lines

View grids with or without grid lines (or with just the outline of the grids displayed).

Colored grid lines

Display grid lines in color (otherwise they are displayed in black).

Hidden line removal + allow examine

View grids with hidden line removal. The **Examine** feature is available only if **Hidden line removal** is selected.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Opacity

Set the opacity for your view. It is functional only if **Hidden line removal** is selected. An opacity of 1.0 means you cannot see inside a grid. An opacity of 0.0 is equivalent to turning hidden line removal off (that is, you can see everything inside the grid).

Enable zoning plot

By default you will only see the parts of a grid that are filled with material. Check this box to see all zoning, including unused elements.

4.3. Settings - Legend

This window lets you set the plot legends you want to display in the View panel.

Color triad

Show a colored triad in the lower left corner of the View panel (otherwise the triad is monochrome).

The axes are color-coded as follows :

Red:	X
Green:	Y
Blue:	Z

Filename visible

Makes the filename (ident) of you model visible in the bottom left corner of the View panel.

Heading visible

Makes the heading of your model visible in the bottom right corner of the View panel.

User heading

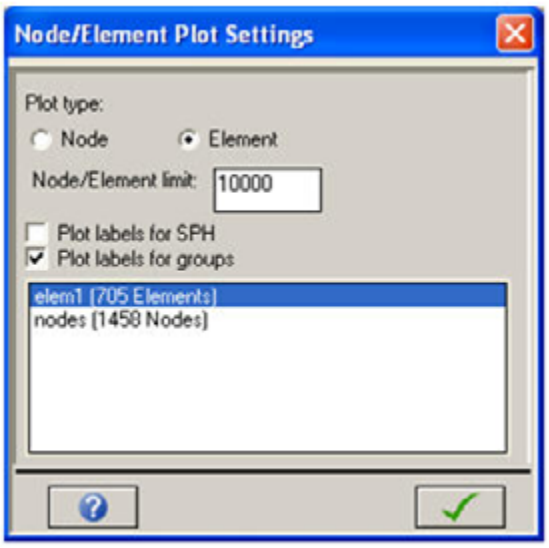
Override the heading of your model with an alternative (user) heading. Enter the user heading in the field provided.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Font size

Enter the size of the font used in the graphics window for the legend, scale, and user heading.

4.4. Settings - Node/Element Numbers



This window lets you change settings for plotting node/element numbers.

Plot type

Plot numbers for nodes or elements.

Node/Element limit

The maximum number of node or element numbers to be plotted. For larger models this number can be increased, which can result in a slower graphic refresh.

Plot labels for SPH

Include SPH particles in the node or element plot.

Plot labels for group

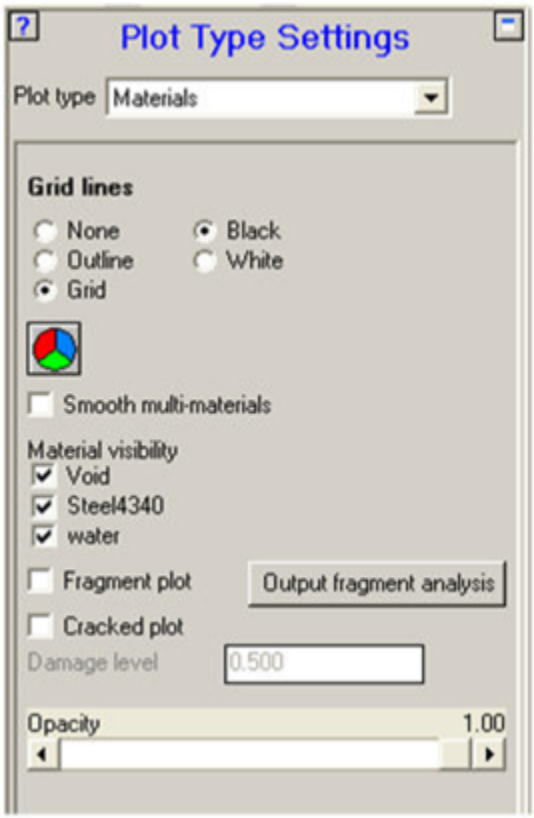
Plot node or element numbers for a particular group. If this option is not selected, node/element numbers are shown for all plotted parts.

Group list

If the **Plot labels for group** is selected, then a list of the defined groups will be shown. Node/element numbers are shown for the selected group.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

4.5. Settings - Materials



This panel lets you set the following options for material plots:

Grid lines

Display grids with or without grid lines (or with just the outline of the grids displayed) and set the color in which the grid lines or outline are drawn.

Clicking this icon lets you set colors for each material using the **Modify Material Color** dialog window.

Smooth multi-materials

Show smooth multi-material interfaces.

Material visibility

The materials you want to be visible in the view.

Fragment plot

View a fragment plot. Fragment plots only show elements that contain material that has not failed.

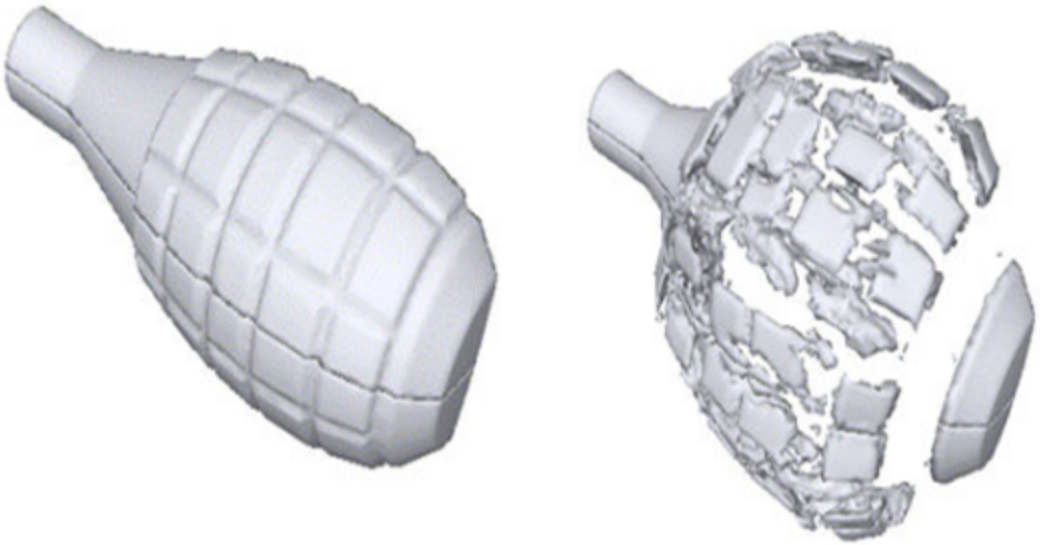
Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Clicking the **Output fragment analysis** button causes fragment data to be output to a file.

The fragment output post processing option is now available for all solvers, including the unstructured solvers.

Note:

The fragment analyzer is supported for restart files only and is disabled for results files. The fragment analyzer is not supported for structured models containing joins. Consider converting the model to unstructured in order to view fragment data.



Simulation of fragmentation of a grenade modeled using unstructured tet elements

Cracked plot

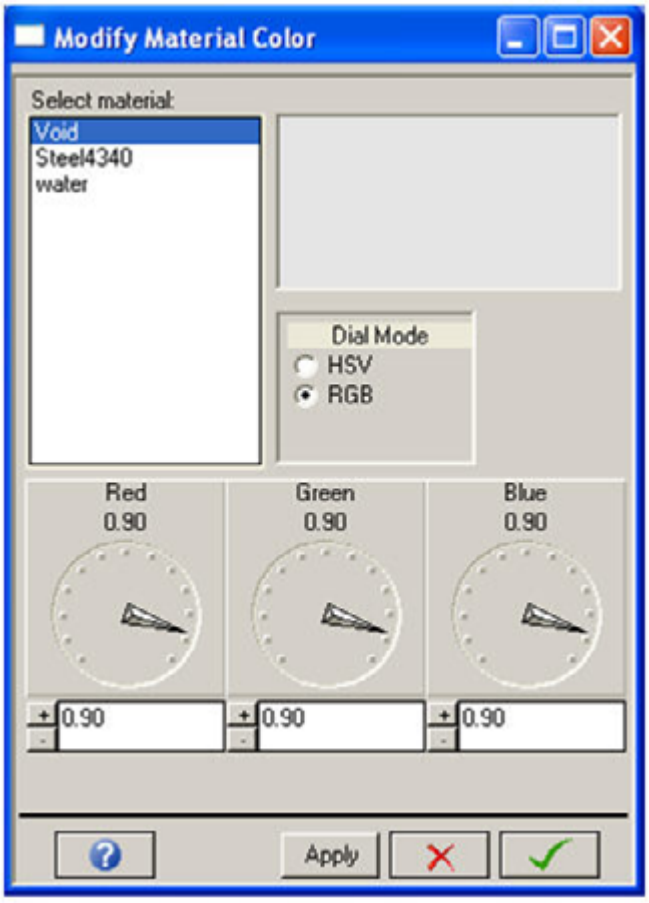
View a cracked material plot. A cracked material plot shows all elements that contain material with damage above the specified **Damage level**.

Opacity

Set the opacity for your view. An opacity of 1.0 means you cannot see inside the grid. An opacity of 0.0 is equivalent to turning hidden line removal off (that is, you can see everything inside the grid).

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

4.5.1. Settings - Materials - Modify Material Color



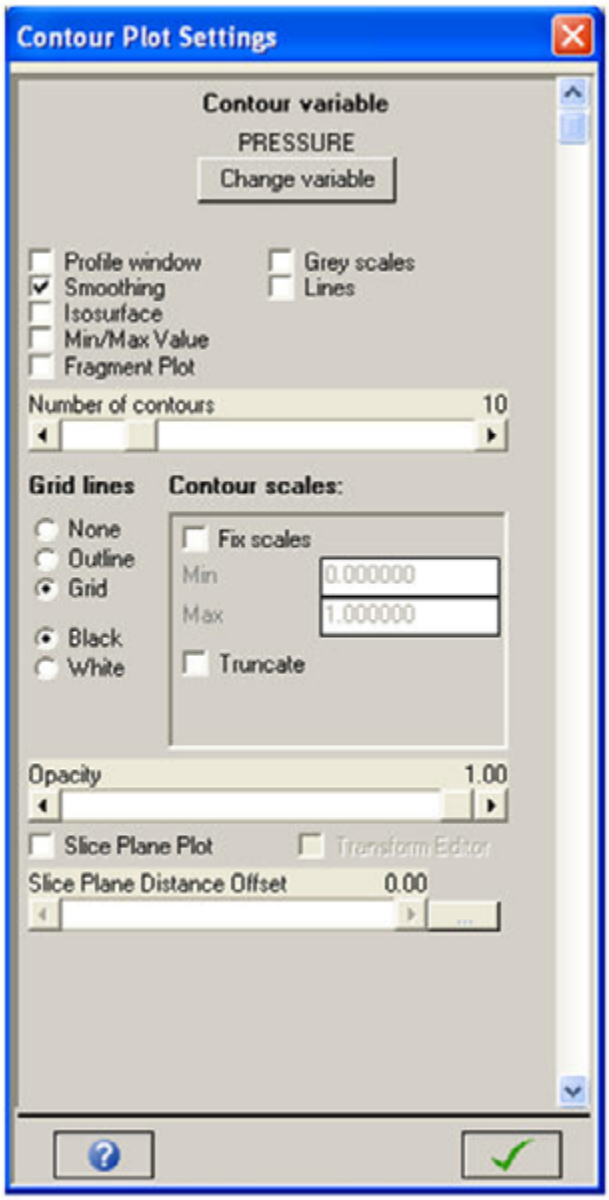
This dialog lets you modify a material color.

1. Select the material whose color you wish set from the list (top left).
2. Choose the dial mode you want to use to set the color:
   - HSV: Hue, Saturation, Value
   - RGB: Red, Green, Blue
3. Use the three dials at the bottom of the window to set the color.

   As you turn the dials, the top right window displays the color that will be used.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

4.6. Settings - Contour



This panel lets you set the following options for contour plots (some options are not present for 2D models):

Contour variable

Click **Change Variable** to open the **Select Contour Variable** dialog window, allowing you to choose the variable you want to contour.

Profile window

Opens the **Profile Plot** dialog window, allowing you to compose and display a profile plot.

This option is only available for structured solvers.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Smoothing

Smooths contour plots. All elements have a single centered value for each variable. If you do not smooth a contour plot each element will be color-coded for the value the variable has in that element. If you smooth a contour plot, then the centered values are interpolated in space to provide smoothly varying values.

Isosurface

View a 3D isosurface plot of the contour variable. You are prompted for the value to be used for the plot.

Min/Max Value

Open a window that displays the minimum and maximum values for the contoured variable.

Grey scales

View contour plots using grey scales instead of color.

Lines

View contour plots with colored lines instead of colored fills.

Number of contours

The number of contours you want to use for your plot.

Grid lines

View grids with or without grid lines (or with just the outline of the grids displayed) and set the color in which the grid lines or outline are displayed.

Contour scales

Usually you should let the maximum and minimum values between which contours are plotted be set automatically using the data in the model. However, you may override these settings in the **Contour scales** box. For example, if a maximum value is entered which is lower than the highest value present in the model, a red area will appear corresponding to all values above the maximum that has been set. Selecting Fix scales lets you fix the current maximum and minimum values. This is particularly useful when you want to generate a slideshow from multiple cycles of a single model. The red area may be removed by selecting **Truncate**.

Opacity

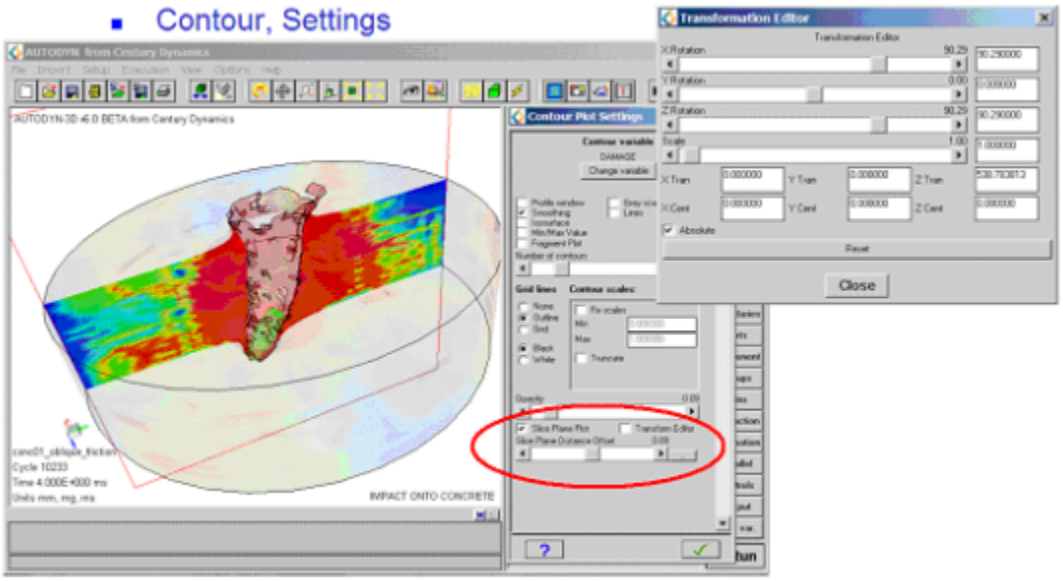
The opacity for your view. An opacity of 1.0 means you cannot see inside the grid. An opacity of 0.0 is equivalent to turning hidden line removal off (that is, you can see everything inside the grid).

Slice Plane Plot

The "Slice Plane" feature is a visual tool to take a section through a model at an arbitrary angle. Results values, as selected under the contour option, are contoured on the selected slice through the model.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

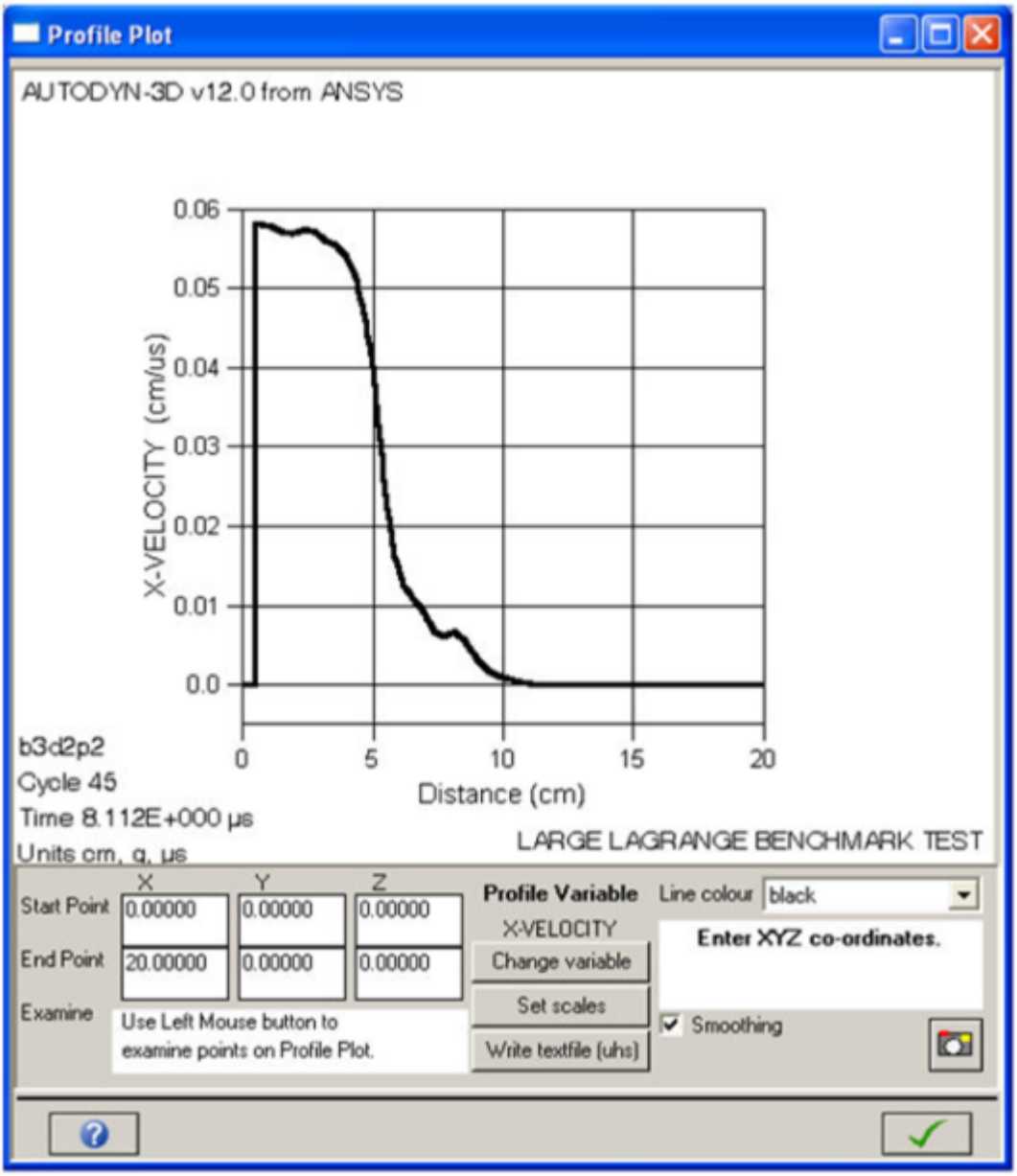
We recommend that you use the slice box feature in conjunction with the Opacity feature of the Contour plot to obtain best results.



Example slice plane plot showing the damage inside a concrete target.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

4.6.1. Settings - Contour - Profile Plot



This dialog window lets you compose and display a profile plot of a chosen variable along some defined line.

This dialog window is only available for structured solvers.

Define the Line along which the variable is profiled

Specify the end points for the line along which the variable is profiled.

You can do this by specifying the coordinates for the start point and end point of the line (lower left).

Alternatively, in 2D, you can select this line interactively in the View Panel using the mouse (hold **Shift**+right mouse button to drag a line across your model).

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Change Variable

Change the variable you want to profile.

Smoothing

Smooth the profile plot.

Line Color

Select the line color for your profile plot.

Write textfile (uhs)

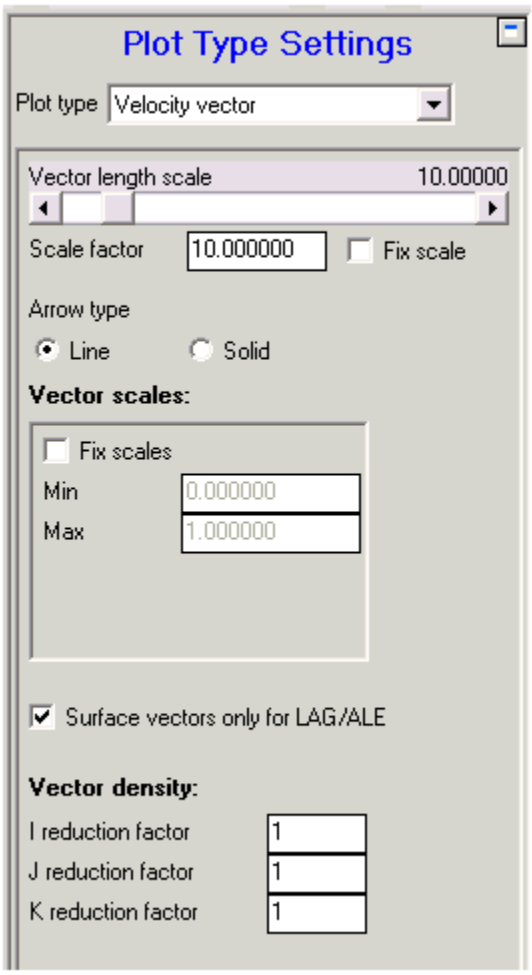
Write digital history data for the current plot to a text file. The text file has an extension `.uhs` and is Microsoft Excel compatible.

Examine

Use the left mouse button to select a point on the graph. The corresponding x and y values are displayed in the Examine panel.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

4.7. Settings - Velocity vector



This panel lets you set the following options for velocity vector plots:

Vector Length Scale

The scale factor of the arrows that are drawn for this plot type. You can use the slider or type in a scale factor in the box below the slider. You can also fix this scale factor so that all plots use the same scale. The range for this factor is calculated from the maximum velocity and the physical dimensions of your model.

Arrow Type

The type of arrows that are plotted. Solid arrows take longer to plot than line arrows (the default).

Vector scales

Autodyn chooses vector scales automatically using the maximum and minimum velocities in the model. You can override these scales and set your own minimum and maximum scale values in this box. You can also fix these scales for all plots (useful when creating slideshows).

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

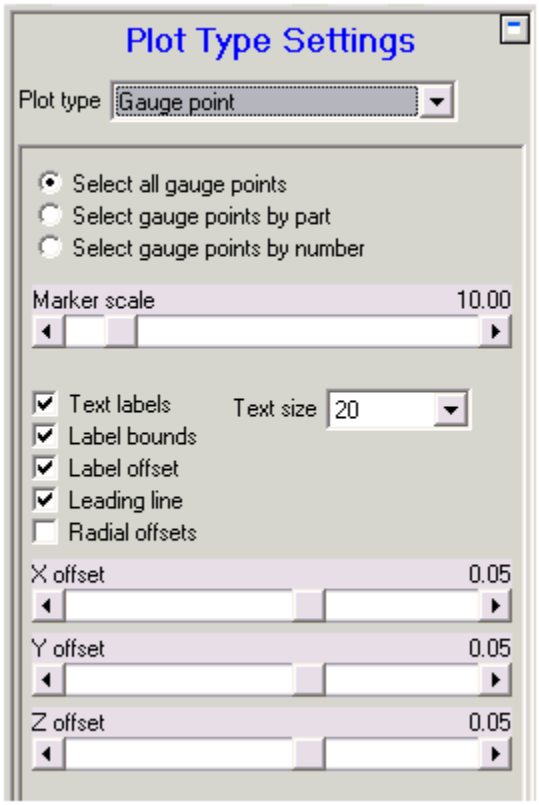
Surface vectors only

Check this box if you want to plot vectors only on material surfaces. This can speed up plotting significantly, but is not always what you need, particularly if your plot uses opacity.

Vector density

Reduces the number of velocity vectors plotted, which can speed up plotting significantly. The option applies only to Lagrange and ALE parts. You specify the reduction factor for each grid index (I, J, and K).

4.8. Settings - Gauge Point



This panel lets you set the following options for gauge point plots:

Select gauge points

These options let you select all gauge points, all gauge points associated with a part, or individually by number. For the last two cases additional dialog prompts you for the part names or gauge numbers as required.

Marker scale

Each gauge point is represented by a marker. This slider lets you set the marker size.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Text labels

Labels each marker with its corresponding gauge number.

Text size

The text size used for the label.

Label bounds

Puts a bounding box around each label.

Label offset

Sets your own offset position for labels.

Leading line

Shows lines joining each marker to its label.

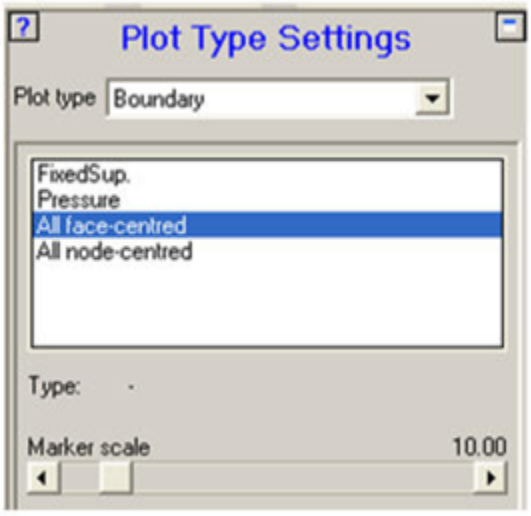
Radial offset

Uses radial label offsets instead of linear offsets.

X / Y / Z offset

These sliders let you adjust the label offset distance in the X, Y, and Z directions.

4.9. Settings - Boundary



This panel lets you set the following options for viewing boundary conditions :

Boundary Condition

Sets the boundary condition to view.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Marker scale

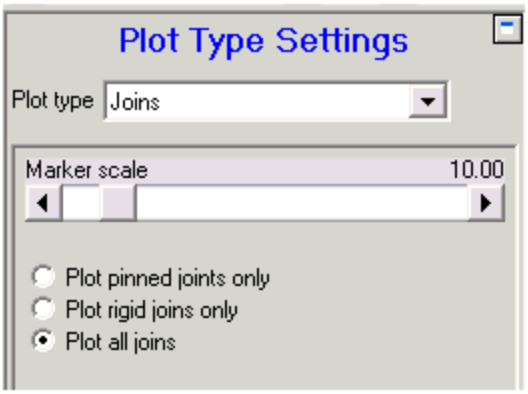
The size of the marker used to show where nodal boundary conditions have been applied.

Note:

For remote boundary conditions:

- Remote Forces and Remote Displacements can be visualized. The scoped nodes for a remote point will be marked on top of the node location where the associated remote point is active.
- If remote boundary conditions are present and if **All node-centred** boundaries are chosen, the view will show markers for all nodes scoped to a remote point, together with the actual remote point itself.

4.10. Settings - Joins



This panel lets you set the following options for viewing joins:

Marker scale

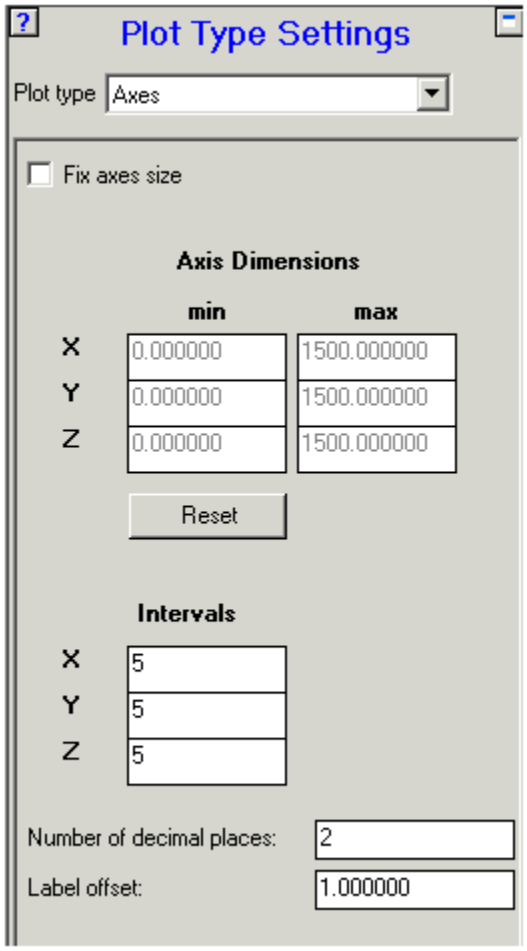
Sets the size of the marker used to show joined nodes.

Joins to plot

Sets the type of joins you want to plot (pinned, rigid, or all joins).

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

4.11. Settings - Axes



Axes are automatically chosen to fit the minimum / maximum range of your model.

This dialog lets you define your own axes:

Fix axes size

Check this box to define your own axes.

Axis Dimensions

Set the min / max range for each of your axes.

Reset

Resets the min / max ranges to those automatically set from your model dimensions.

Intervals

Set the number of intervals you want displayed on each of your axes.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

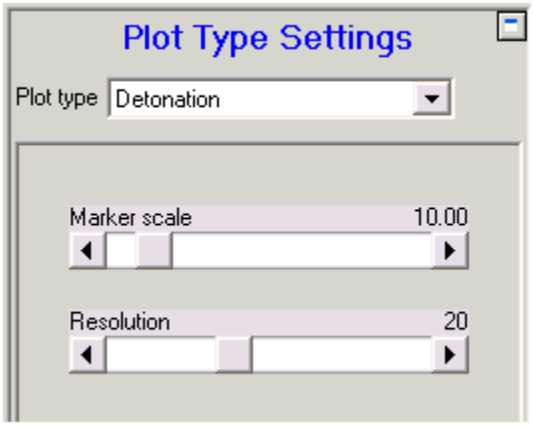
Number of decimal places

The number of decimal places to be displayed for the axes.

Label offset

Set the label offset to control the distance between the axes and the labels.

4.12. Settings - Detonation



This panel lets you set the following options for viewing detonation points:

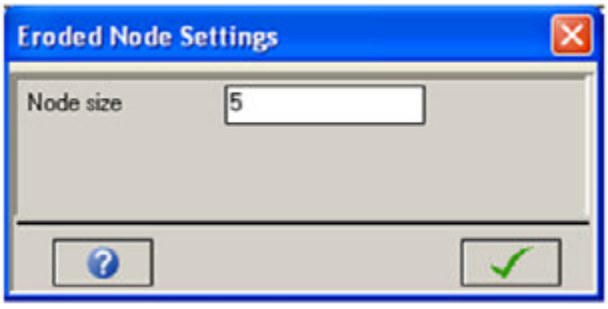
Marker scale

Sets the size of the marker used to show detonation points.

Resolution

Sets the marker resolution.

4.13. Settings - Eroded Nodes



This window lets you set the node size for plotting eroded nodes.

Enter an integer number; the higher the number, the larger the node.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

4.14. Settings - Unused Regions

This window lets you set options for the plotting of unused regions.

Grid lines

Select whether you want to plot the outline or grid for unused zones and whether the outline/grid is drawn in black or white.

Plot contours

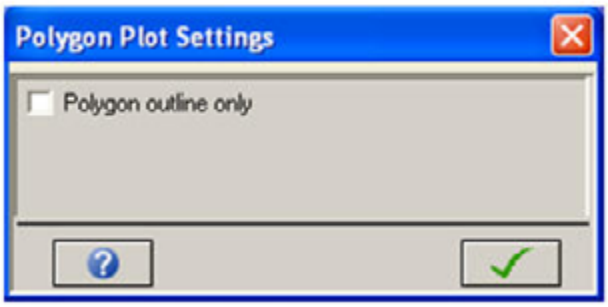
Check this box if you want the contour variable contoured on unused faces (for contour plots). Otherwise the face is displayed in grey.

Opacity

Set the opacity for unused regions. An opacity of 1.0 means you cannot see inside the region. An opacity of 0.0 is equivalent to turning hidden line removal off (that is, you can see everything inside the region).

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

4.15. Settings - Polygons

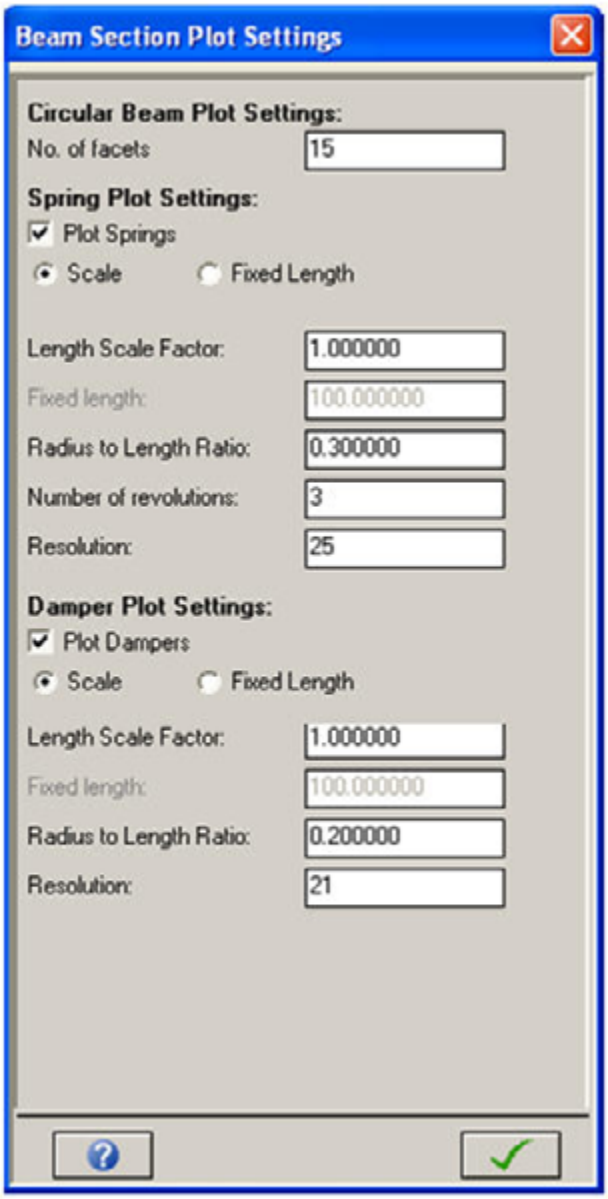


This window lets you turn on the plotting of polygons (used in 2D Euler-Lagrange coupling).

Check the box to turn outline plotting on.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

4.16. Settings - Beam Sections



This window lets you control the plotting of beams.

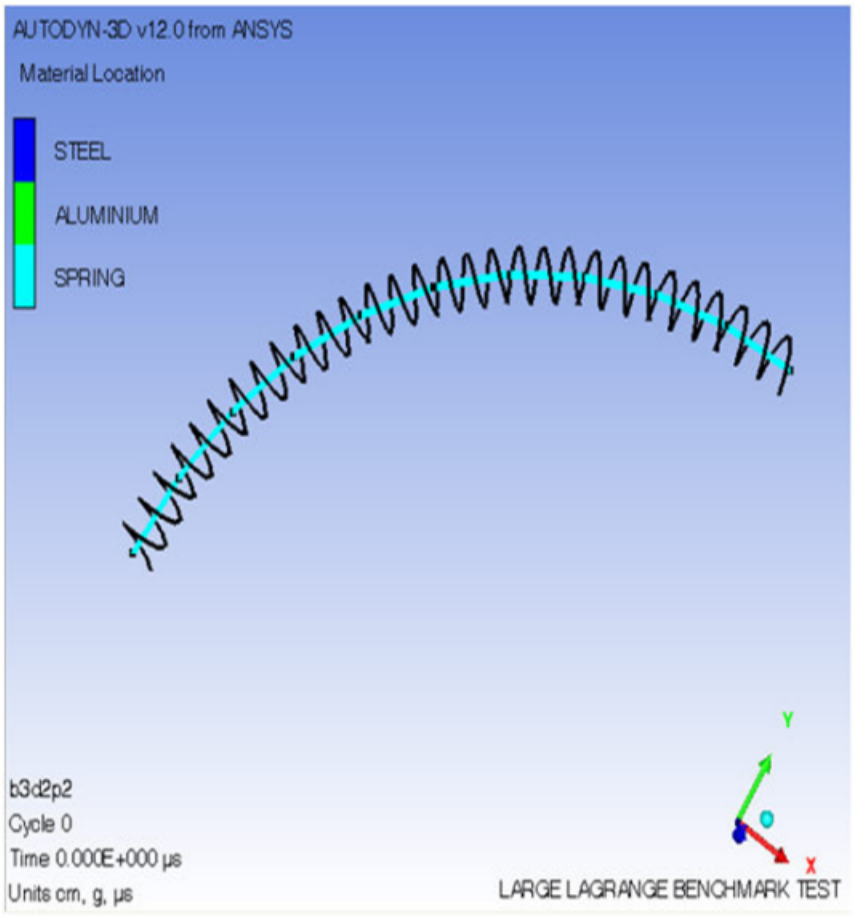
Circular Beam Plot Settings

The number of facets to be plotted for each circular cross-section beam.

Spring Plot Settings

- Select the Plot Springs toggle to enable the spring plot.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Scale

Normalizes each plotted damper with the length of the damper element.

Fixed Length

Displays each plotted damper as a fixed size.

Length Scale Factor

The Length Scale Factor is only used together with the scale option. If a length scale factor of one is chosen, each damper will be displayed as half the length as the element.

Fixed Length

If the fixed length option has been selected, specify the length of each plotted damper in this field.

Radius to Length Ratio

Sets the radius to length ratio of each plotted spring.

Number of revolutions

Sets the number of revolutions for each plotted spring.

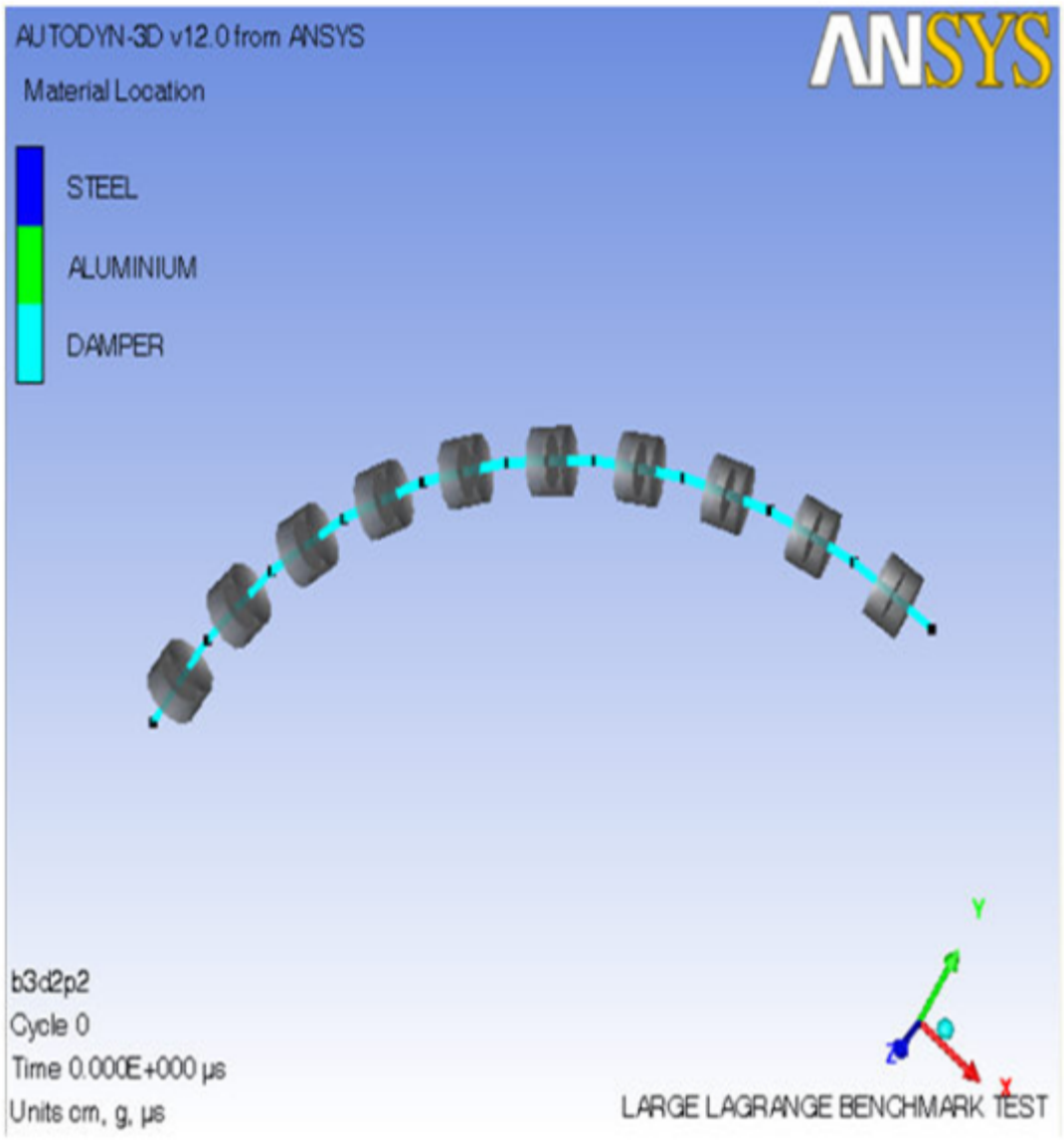
Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Resolution

Sets the number of lines used to represent each plotted spring.

- **Damper Plot Settings**

Select the Plot Damper toggle to enable the damper plot.



Scale

Normalizes each plotted damper with the length of the damper element.

Fixed Length

Displays each plotted damper as a fixed size.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Length Scale Factor

The Length Scale Factor is only used together with the scale option. If a length scale factor of one is chosen, each damper will be displayed as half the length as the element.

Fixed Length

If the fixed length option has been selected, specify the length of each plotted damper in this field.

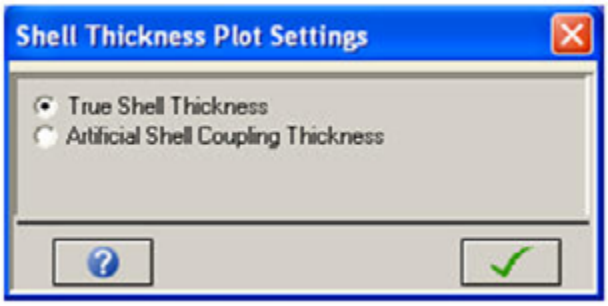
Radius to Length Ratio

Sets the radius to length ratio of each plotted damper.

Resolution

Sets the number of lines used to represent each plotted damper.

4.17. Settings - Shell Thickness

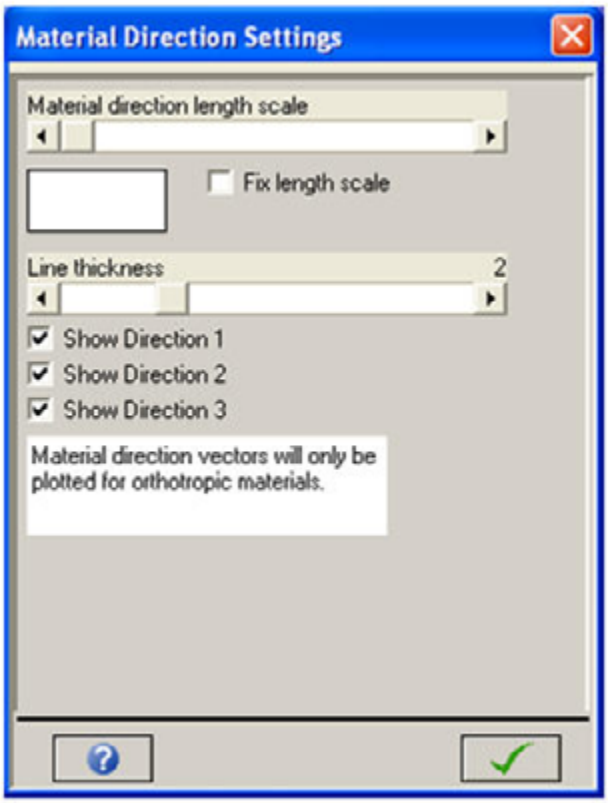


This window lets you select the thickness you want to use for rendering shells.

You can use either the true shell thickness or the artificial thickness that is used for coupling shells to Euler.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

4.18. Settings - Material Direction



This window lets you change settings for plotting material directions.

Material direction length scale

Varies the length of the lines displaying the local axes.

Fix length scale

Fixes the local axes length scale.

Line thickness

Varies the thickness of the lines displaying the local axes.

Show Direction 1

Displays the local axis for direction 1.

Show Direction 2

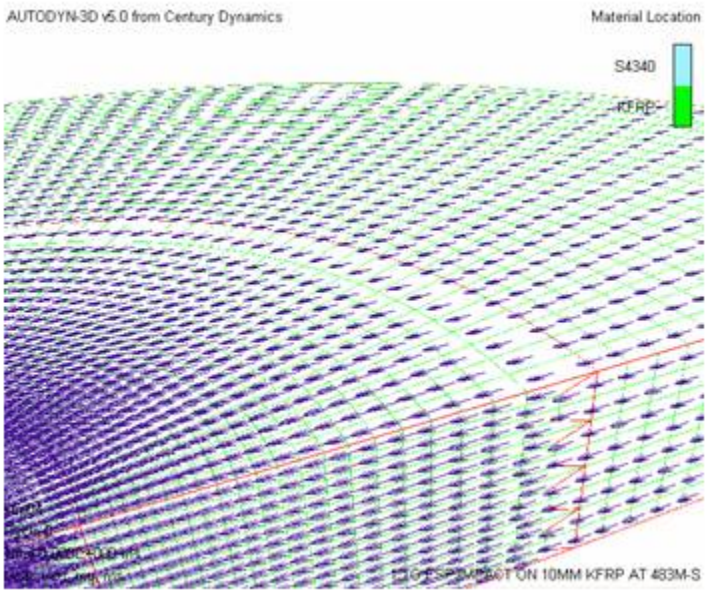
Displays the local axis for direction 2.

Show Direction 3

Displays the local axis for direction 3.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

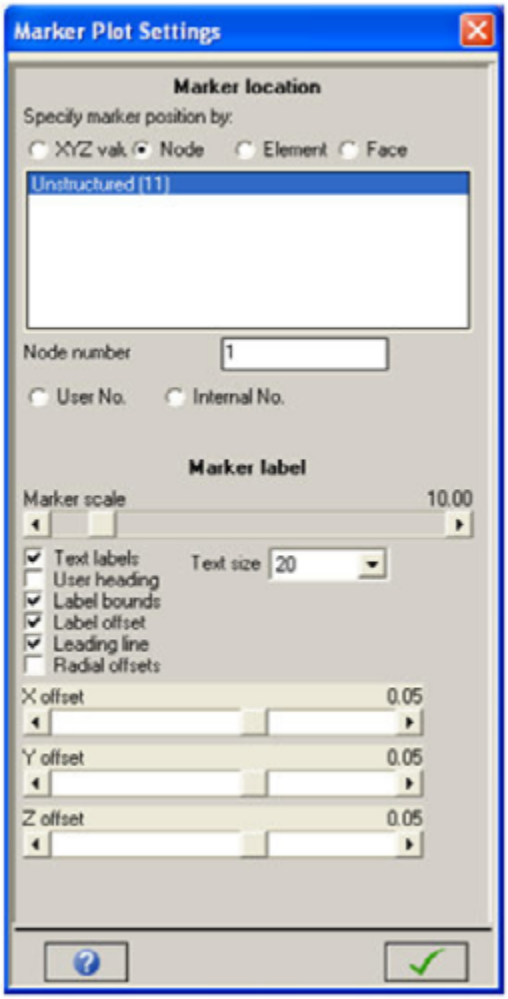
For large models containing mainly volume elements, you will get better results by selecting the **Settings** > **Display** > **Surface** rendering option.

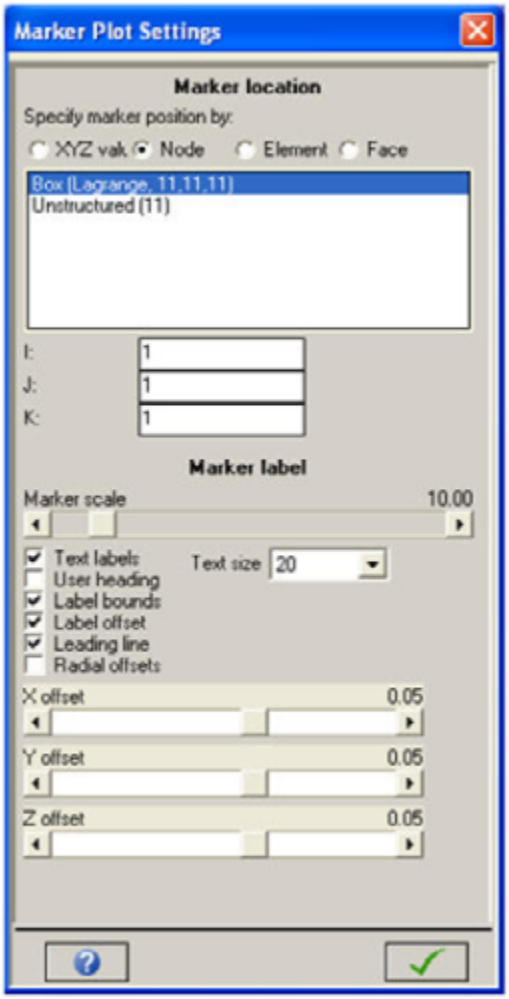


For unstructured shells, the material direction plot is only available for non cycle-zero files.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

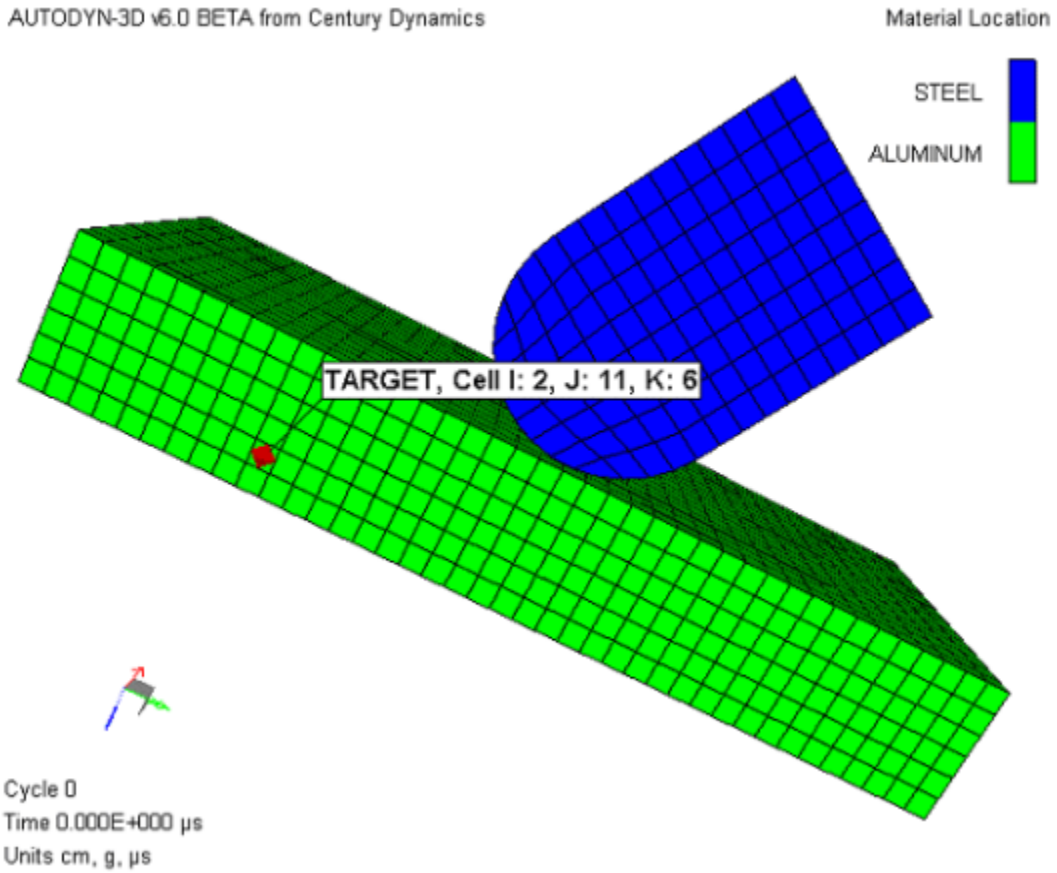
4.19. Settings - Marker





The Marker Plot lets you highlight a particular position, node, or element of interest.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Specify Marker Position

Choose to specify the marker position either by XYZ coordinates or by specifying a part and I/J/K values for a node or element.

XYZ coordinates

The coordinates of the marker location.

Node/Element/Face

Highlights a particular node, element or face of an element.

Parts List

The part containing the node, element, or face to be highlighted.

IJK

Select the I, J, and K values for the node, element or face of interest in case the part selected is structured.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Node Number/Element Number/Face Number

Enter the node, element for face number of interest in case the part selected is unstructured. Depending on the selection of User No. or Internal No. the node, element or face number displayed is the original User Number or the Autodyn internal number.

Text labels

Labels each marker.

Text size

The text size used for the label.

Label bounds

Puts a bounding box around each label.

Label offset

Sets your own offset position for labels.

Leading line

Shows lines joining each marker to its label.

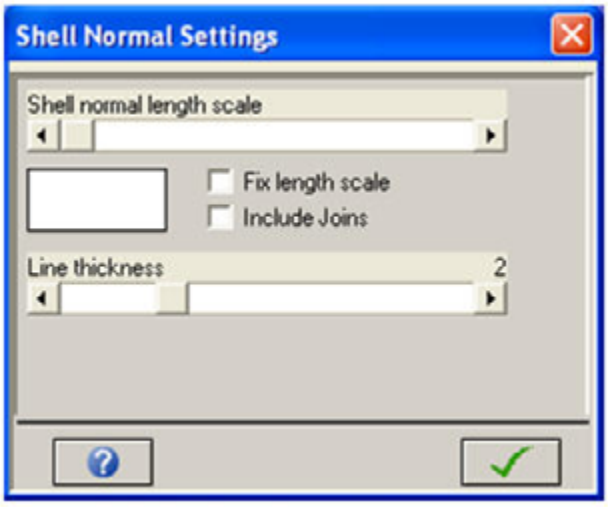
Radial offset

Check this box if you want to use radial label offsets instead of linear offsets.

X / Y / Z offset

These sliders let you adjust the label offset distance in the X, Y, and Z directions.

4.20. Settings - Shell Normal



This window lets you change settings for plotting shell normals.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Local Axes length scale

Varies the length of the arrows displaying the shell direction.

Fix length scale

Fixes the shell normals length scale.

Include Joins

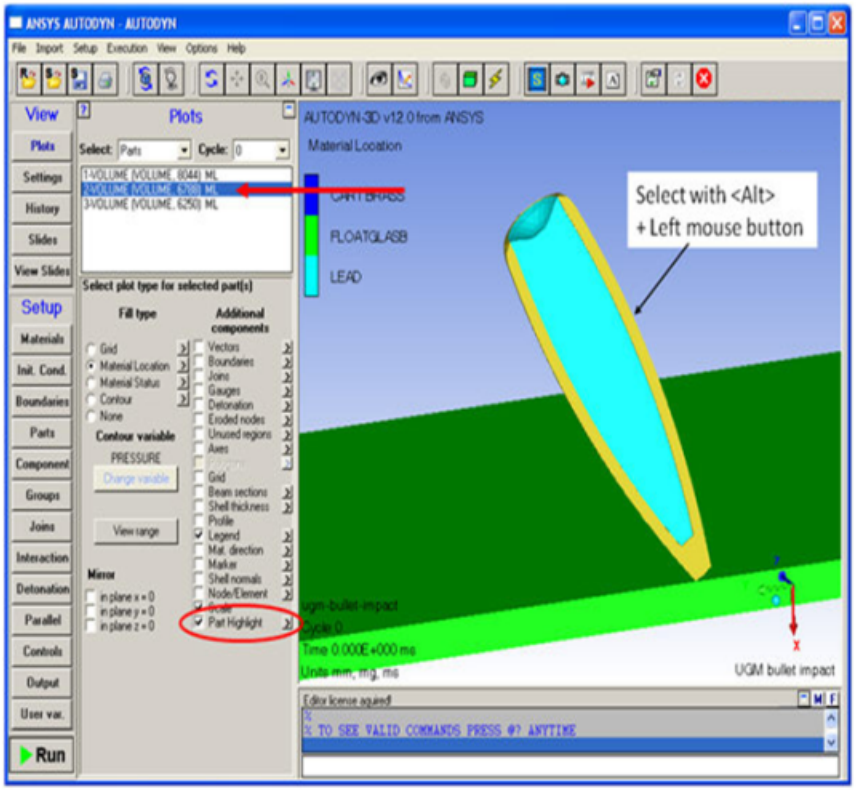
Select this toggle to show an average normal for joined nodes. If this toggle is unselected, a normal will be shown for each of the nodes joined.

Line thickness

Varies the thickness of the lines displaying the shell normals.

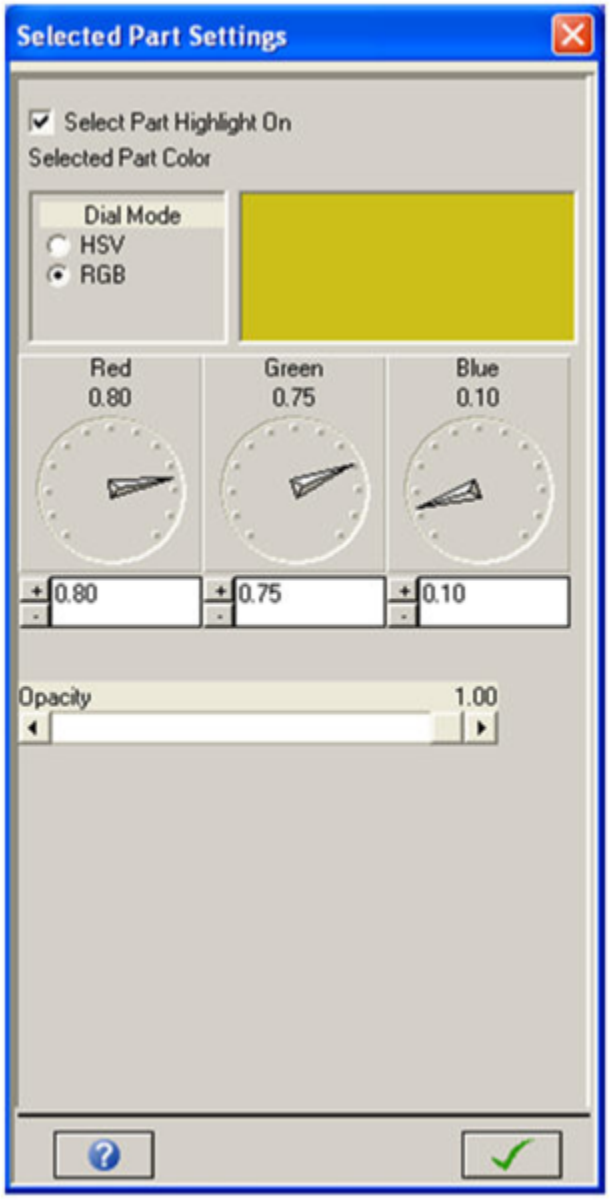
4.21. Settings - Part Highlights

When this option is selected, Parts selected in the plots panel will be highlighted on the displayed model. Selecting multiple parts in the parts list will highlight all the selected Parts.



Using **Alt**+left mouse button you can select a part in the model view window and make it the current part. The part selected with the mouse will be selected in either the part list in the plots panel or parts panel. This is useful to identify parts and to make a particular part the current part active for zoning, filling or other part operations.

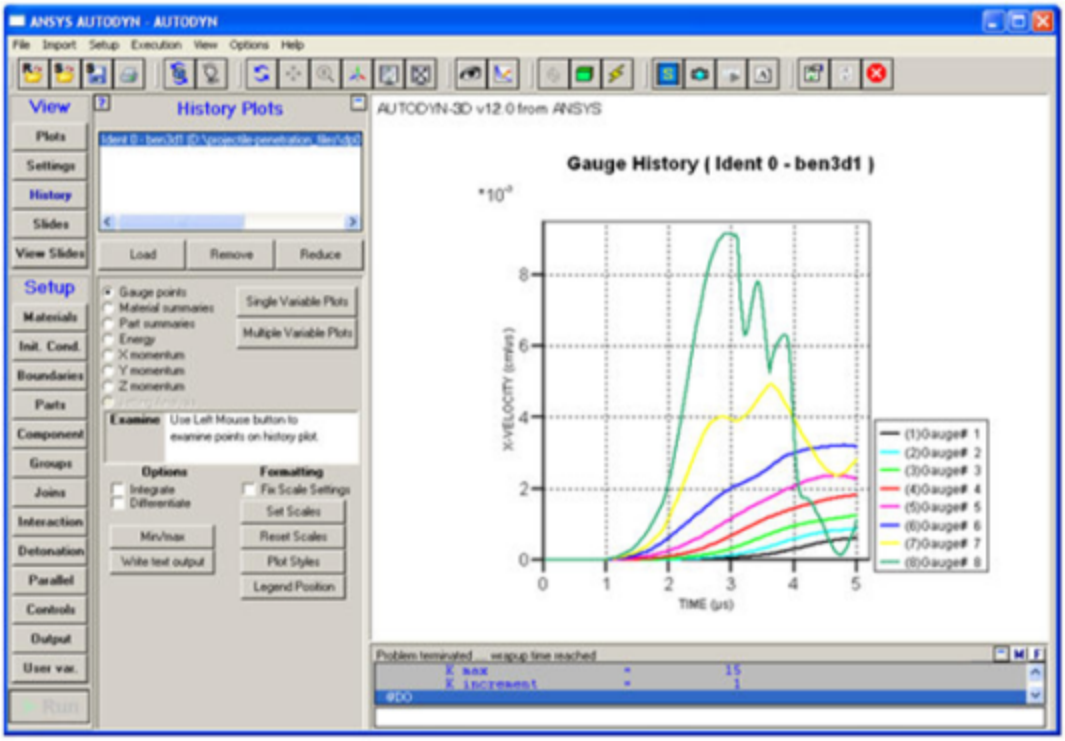
Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



The default highlight color is golden yellow; you may alter the color and opacity of the highlight color that is applied to selected parts using the associated options panel.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Chapter 5: Autodyn History



This panel lets you compose and display time histories of Gauge points and Summary variables.

When you select this panel, the history and summary data associated with the model you are working with will be automatically loaded and the model's Ident will be displayed in the window at the top of the panel.

Load

Load history data for other models.

Remove

Remove history data for the selected model (at least one model must remain loaded at all times).

Reduce

Reduce the amount of history data stored for a model.

History Type

Select the type of history you want to view from the following options:

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Gauge Points

Variable histories at gauge points.

Material Summaries

Mass, energy and momentum sums for each material.

Part Summaries

Mass, energy, and momentum sums for each Part.

Energy

Energy balance.

X momentum

X momentum balance.

Y momentum

Y momentum balance

Z momentum

Z momentum balance

Jetting Analysis

Jetting analysis plots (active only if model uses Jetting).

Single Variable Plots

Compose plots of a single variable

Multiple Variable Plots

Compose plots including multiple variables

Examine

Use the left mouse button to select a point on the graph. The corresponding curve label and x and y values are displayed in the Examine panel.

Integrate

View integrals of the variables in the current plot.

Differentiate

View differentials of the variables in the current plot.

Min/Max

View maximum and minimum values for the current plot.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Write textfile (uhs)

Output the data from the current plot (Write UHS for current history plot) or output all the gauge data from a history file (Write Output for all gauges and variables in selected ident).

Set scales

Autodyn automatically chooses scales for your plots based on minimum and maximum values of the data. Clicking this button lets you set your own scales.

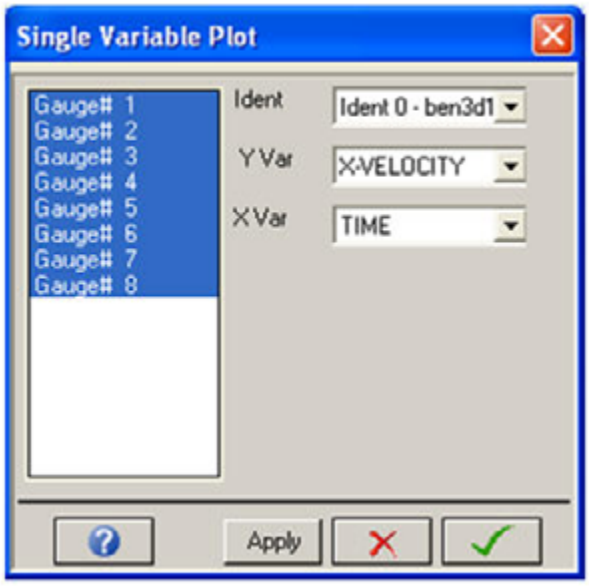
Plot Styles

Set the color, style and thickness of lines for each plot.

Legend Position

Position the legend anywhere in the view.

5.1. History - Single Variable Plot



This window lets you compose a single variable plot.

Single variable plots can contain one or more plots (histories) for the same X and Y variables.

Ident

Select an Ident from the models you have loaded.

Y Var

The variable assigned to the Y axis.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

X Var

The variable assigned to the X axis (most often TIME).

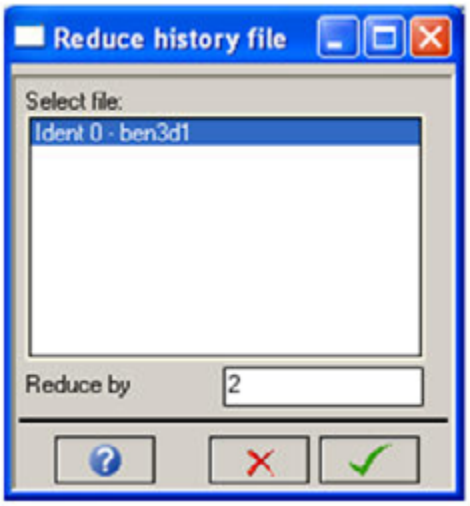
Histories to display

From the list on the left, choose as many histories as you wish to display in the plot (use the **Ctrl** and **Shift** keys to make your selection).

Important:

The choices you have available for variables and histories will depend on the type of plot you have chosen (Gauge points, Material Summaries, and so on).

5.2. History - Reduce



It is usual to save history data very frequently (the default is every cycle) in order to capture high frequency responses.

If a calculation is run for a large number of cycles the history files can become very large.

This window lets you reduce the size of history files by removing data for some cycles.

Select file

The ident of the file you wish to reduce in size.

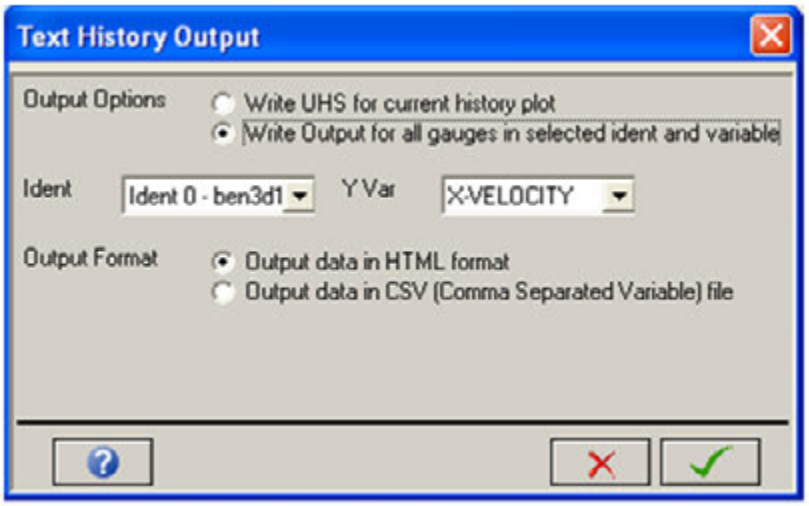
Reduce by

Specify the reduction factor in this field.

- If you specify a reduction factor of 2, data for every other cycle will be removed from the file.
- If you specify a reduction factor of 3, data for two out of every three cycles will be removed from the file.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

5.3. History - Write Text Output



This dialog lets you output the data from the current plot (**Write UHS for current history plot**) or output all the gauge data from a history file (**Write Output for all gauges and variables in selected ident**).

Output Options

Write UHS for current history plot

Output the data from the currently plotted history file to a CSV (comma separated variable) file, which is suitable for importing into Microsoft Excel.

Write Output for all gauges and variables in selected ident

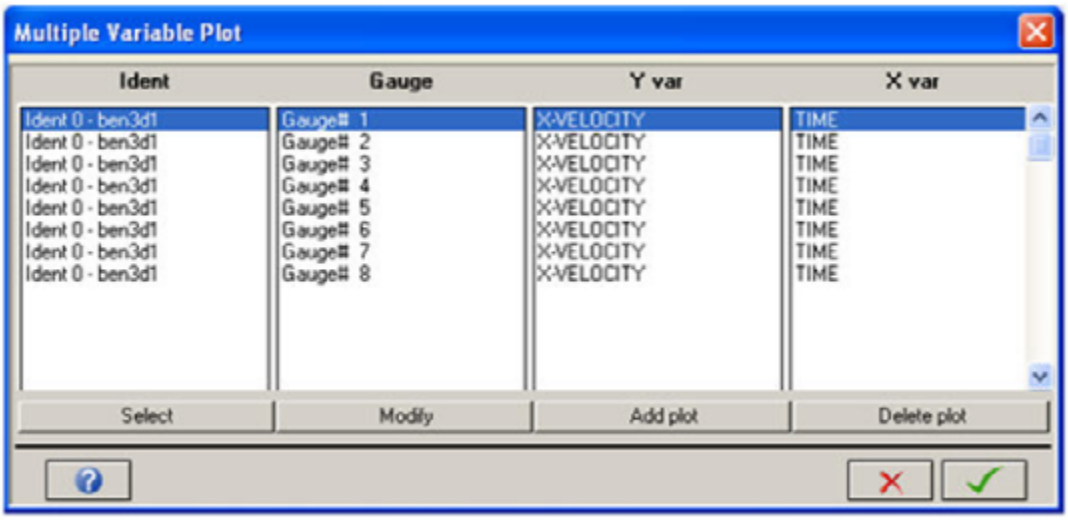
Selecting an **Ident** and variable to output, all the data for each gauge for the selected variable and ident is written to the output file.

Output Format

Select the format for the data output, either a CSV (Comma separated variable) file or an HTML file.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

5.4. History - Multiple Variable Plot



This window lets you compose a multiple variable plot.

Multiple variable plots can contain one or more plots (histories), each with different variables for the X and Y axes.

This window list the plots (histories) you have currently define, displaying for each one its Ident, Gauge number, X variable and Y variable.

Below this list are four buttons:

Select

Click this button to quickly add multiple plots to the list.

Modify

Select a plot in the list and click this button to modify its settings.

Add plot

Select a plot in the list and click this button to add a new plot to the list. The initial settings for the new plot will be the same as the selected plot, but you can modify these values immediately in the dialog window that opens.

Delete plot

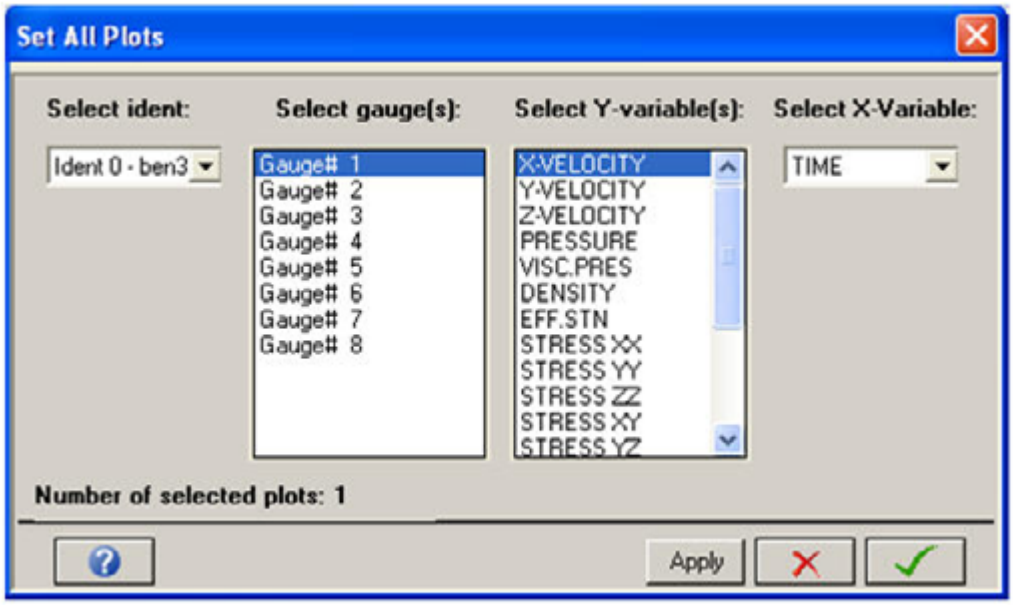
Click this button to delete a plot from the list.

Important:

The choices you have available for gauges and variables will depend on the type of plot you have chosen (Gauge points, Material Summaries, and so on).

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

5.4.1. History - Multiple Variable Plot - Set All Plots



This window lets you quickly add multiple plots to a multiple variable plot list.

Select ident

Choose an ident from the models you have loaded.

Select gauge(s)

Choose one or more gauges (use the **Ctrl** and **Shift** keys in the usual way to make your selection).

Select Y-variable(s)

Choose one or more variables for the Y axis (use the **Ctrl** and **Shift** keys in the usual way to make your selection).

Select X-variable

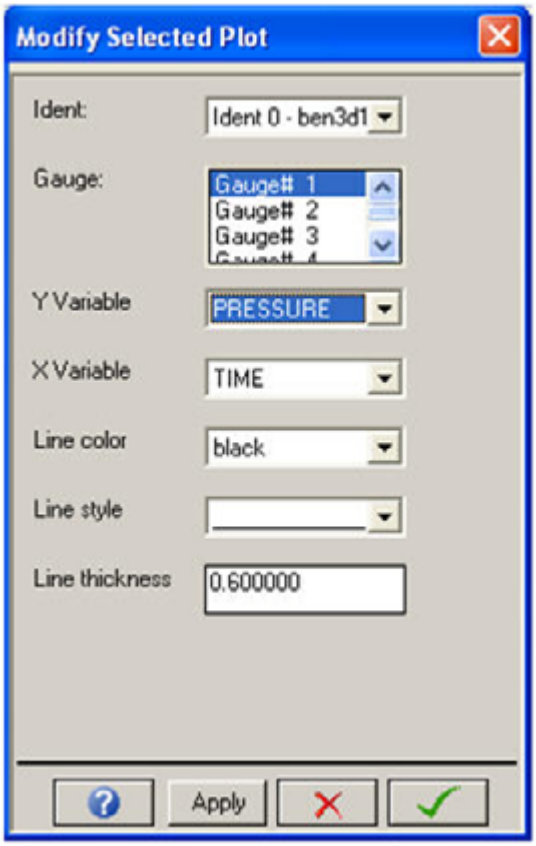
Choose a variable for the X axis.

Number of selected plots

How many plots you have defined with your current selections. The number of plots is the union of all selections in the four input regions.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

5.4.2. History - Multiple Variable Plot - Modify Selected Plot



This window lets you set or modify parameters for a plot to be included in a multiple variable plot.

Ident

The Ident for the plot from the models you have loaded.

Gauge

The gauge to be plotted

Y Variable

The Y Variable

X Variable

The X Variable

Line color

The color of the line.

Line style

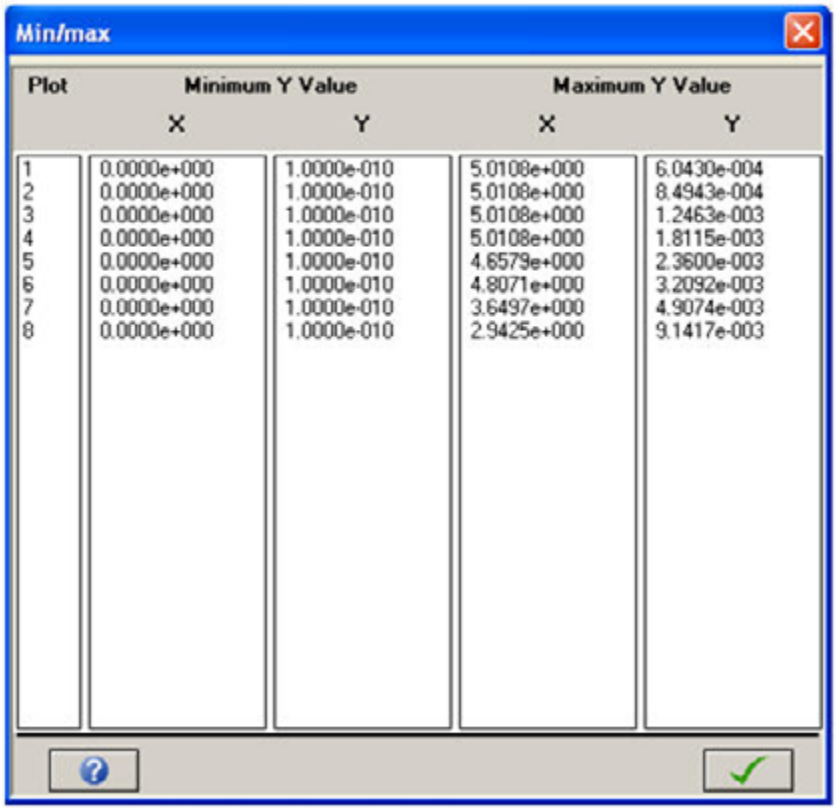
The line style (solid, dashed, etc.)

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Line thickness

The line thickness.

5.5. History - Min/max



This window displays the maximum and minimum values for each history plot shown in the current view.

X min is the X value at which the minimum Y value occurs

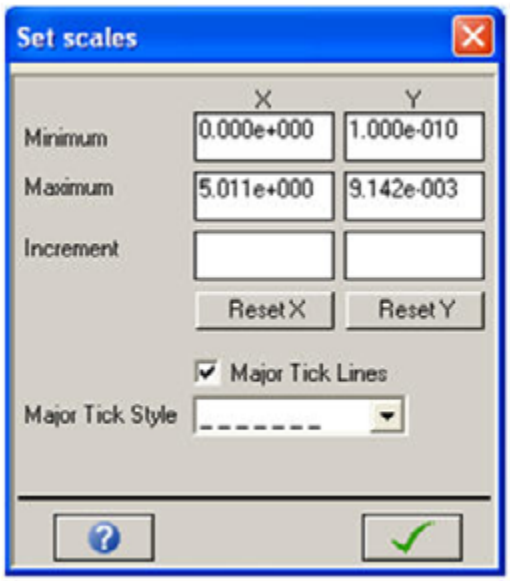
Y min is the minimum Y value

X max is the X value at which the maximum Y value occurs

Y max is the maximum Y value

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

5.6. History - Set Scales



This window lets you set your own scales for history plots.

Minimum

The minimum values for your X and Y scales

Maximum

The maximum values for your X and Y scales

Increment

The X and Y increments where tick marks will be placed on the scales.

Reset X

Resets the maximum and minimum X values to those calculated automatically by Autodyn.

Reset Y

Resets the maximum and minimum Y values to those calculated automatically by Autodyn.

Major Tick Lines

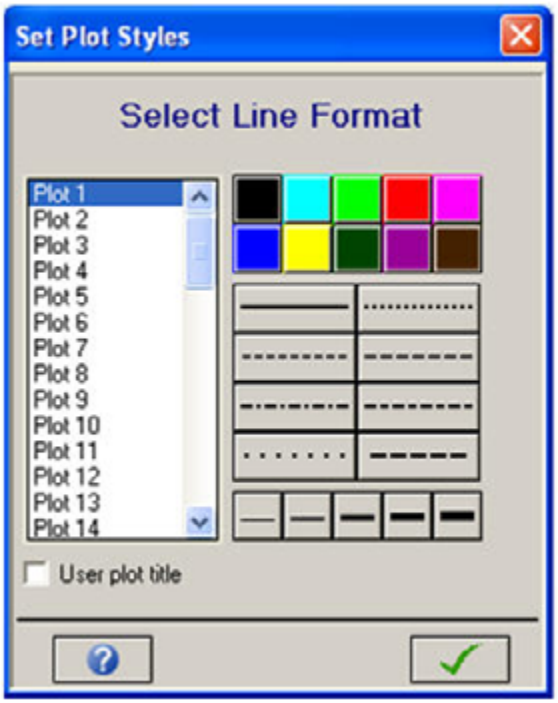
Check this box if you want major tick lines to extend from left to right and top to bottom across your plot.

Major Tick Style

The line style used to draw tick lines (solid, dashed, etc.)

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

5.7. History - Set Plot Styles



This window lets you set the color, style, and thickness of lines for each plot

Plot Number

Plots are numbered in the order you specify them, starting at 1. The box on the left lets you select a plot number whose style you wish to set.

Color

Select a color for the selected plot from the top two rows on the right.

Line Style

Select a line style for the selected plot from the next four rows on the right

Line Thickness

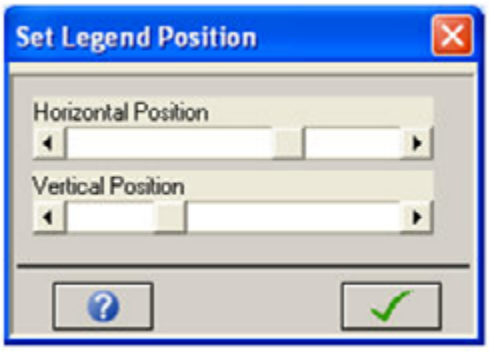
Select a line thickness for the selected plot from the bottom row on the right

User plot title

Check this box if you want to supply your own title for the plot. The default title is "History Plot".

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

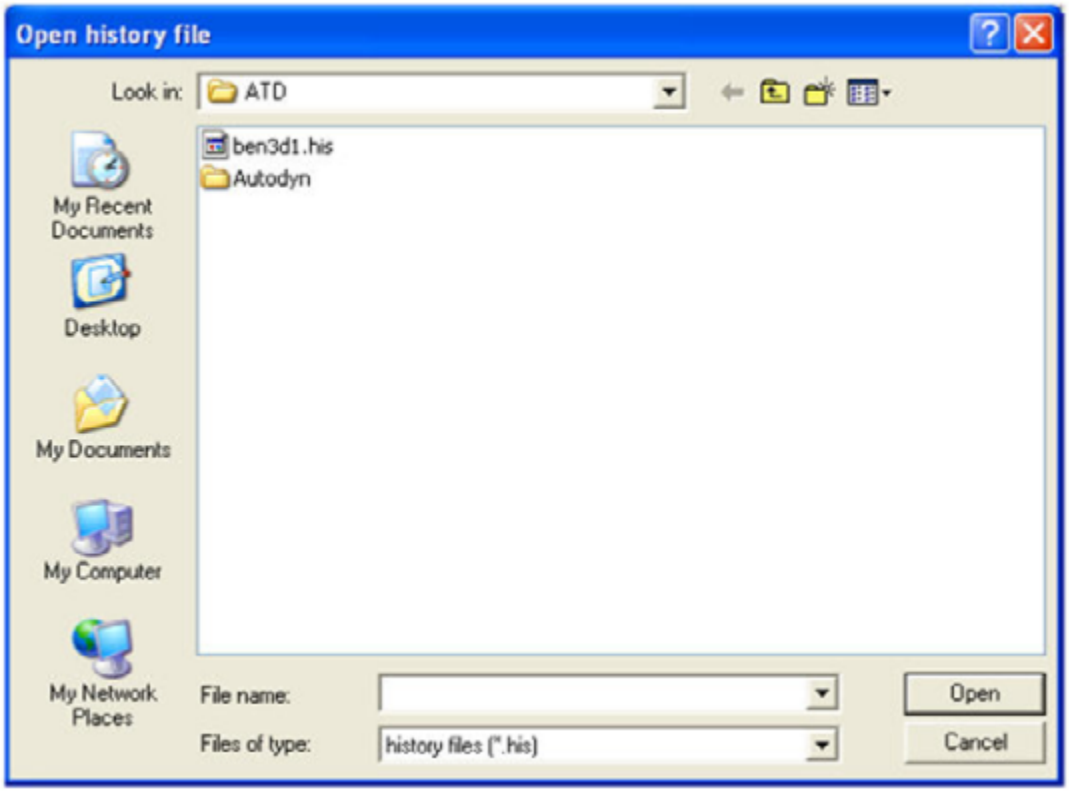
5.8. History - Set Legend Position



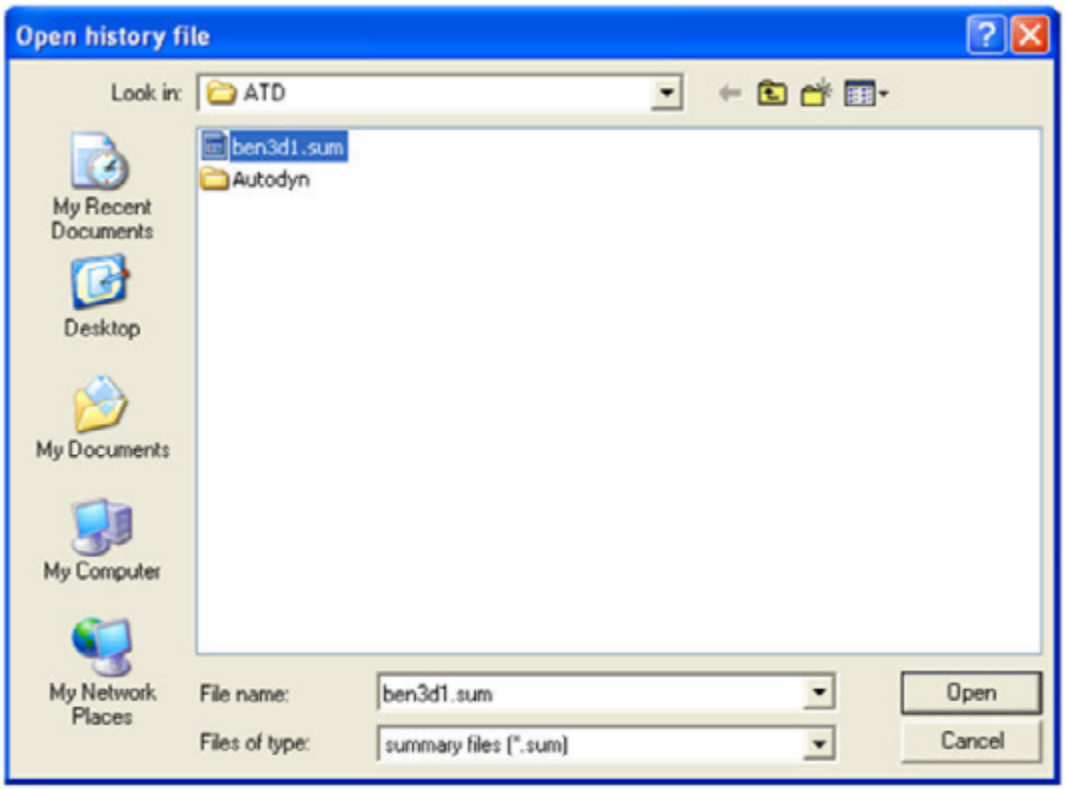
This window lets you place the plot legend anywhere in the view.

Use the two sliders to adjust the horizontal and vertical position of the legend until it's where you want it.

5.9. History - Load



Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



This window lets you load history files for plotting.

History data is stored in two types of files:

.his

These files contain history data for the gauge points defined for a model.

.sum

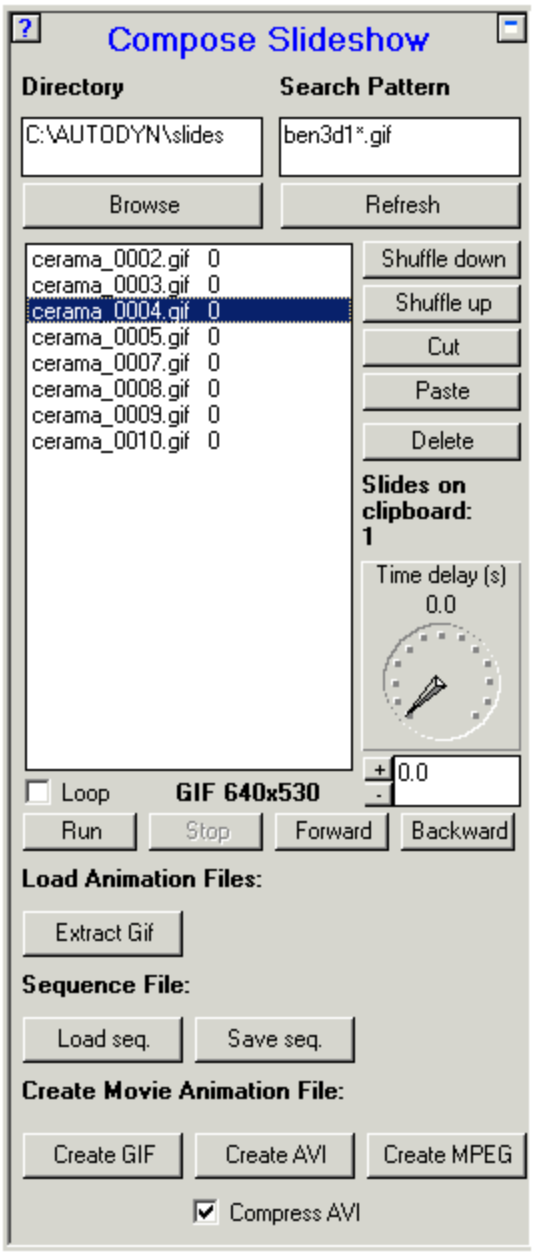
These files contain history data for summary plots (Material Summaries, Part Summaries, Energy, and so on).

Depending on the plot type you have chosen, this window lets you navigate your file system to find `.his` or `.sum` files for loading.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information
of ANSYS, Inc. and its subsidiaries and affiliates.

Chapter 6: Autodyn Slides



This panel lets you compose slide shows from images created using the **Text Slide**, **Camera** (single slide) and **Animation** (multiple slides) buttons on the toolbar.

The left center box shows the current slide sequence. When you select a file in the slide sequence, the image of that file is automatically displayed in the View panel. Initially the slide sequence will contain all the files in the current **Directory** that satisfy the **Search Pattern**, listed in alphanumeric order.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Directory

The current directory where the slides are stored and processed is shown in this Box.

Click the **Browse** button below the box to change this directory.

After changing the directory, click the **Refresh** button to update the list of slides in the slide sequence box. You can also click the **Refresh** button if you want to abandon your current sequence and start over with all files in alphanumeric order.

Search Pattern

You can type a search pattern in this field that restricts the files you see in the slide sequence box.

In the example above the search pattern restricts files you see to those having an ident "ben3d1" and extension **.gif**.

All **.gif**, **.bmp**, **.tiff** and **.jpeg** files are shown if the search pattern does not include an extension.

Shuffle Up/Down

These buttons let you move one or more files up or down in the slide sequence.

Select the group of files you want to move (using the **Shift** key to select more than one file), and click the **Shuffle Up** or **Shuffle Down** button to move them collectively one position up or down respectively.

Cut

Select one or more files in the slide sequence and click this button to remove them. The removed files are placed on the clipboard and you can use the **Paste** button to restore them at some other point in the sequence. If you use **Cut** a second time, the clipboard is cleared and the newly removed files are placed on it.

Paste

Inserts all files from the clipboard **after** the selected file in the slide sequence.

Slides on clipboard

Displays how many slide files are currently on the clipboard.

Time delay

Select one or more files in the slide sequence and rotate the dial to set the time delay you want to apply to them (a slide's time delay is the time, in seconds, for which it is shown).

Instead of using the dial, you can alternatively type the time delay in the field below the dial.

Loop

Check the box if you want your slide show to replay automatically.

Run

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Plays the current slide sequence. The entire sequence is played, irrespective of which files are currently selected.

Stop

Stops the playing of the current slide sequence.

Forward / Backward

Let you quickly change your selection to the preceding (**Backward**) or following (**Forward**) file in the slide sequence.

Since this procedure automatically updates the View panel, this is a good way of viewing your slides manually.

Extract GIF

Extracts individual `.gif` files from an animated GIF and places them in the current directory.

> **Warning:**
>
> This option removes the current slide sequence and replaces it with a sequence containing only the newly extracted files (irrespective of the current search pattern).

Save Seq.

Stores the current slide sequence in a sequence file.

Load Seq.

Loads a sequence stored in a sequence file and applies it to the current directory.

> **Warning:**
>
> If the files listed in the sequence do not exactly match those in the current directory, results may be unpredictable.

Create GIF

Creates a GIF animation from the current slide sequence.

Create AVI

Creates an AVI animation from the current slide sequence.

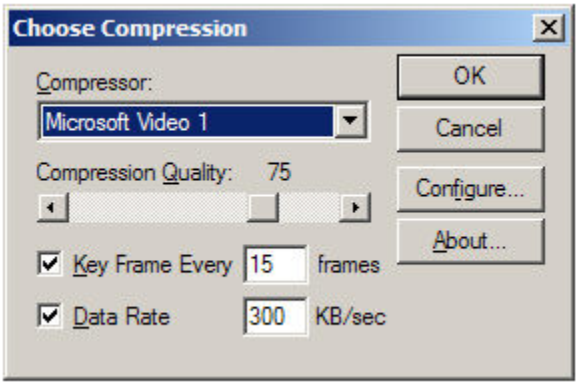
Create MPEG

Creates an MPEG animation from the current slide sequence.

Compress AVI

Select this checkbox to choose compression for AVI.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



In the **Choose Compression** dialog, select the type of compression to use for the movie. The size and quality of the resulting AVI animation will depend on the choices made in this panel.

6.1. Slides Toolbar

The following operations are available:

6.1.1. Slides Setup
6.1.2. Capture Current Image
6.1.3. Capture Sequence
6.1.4. Text Slide

6.1.1. Slides Setup

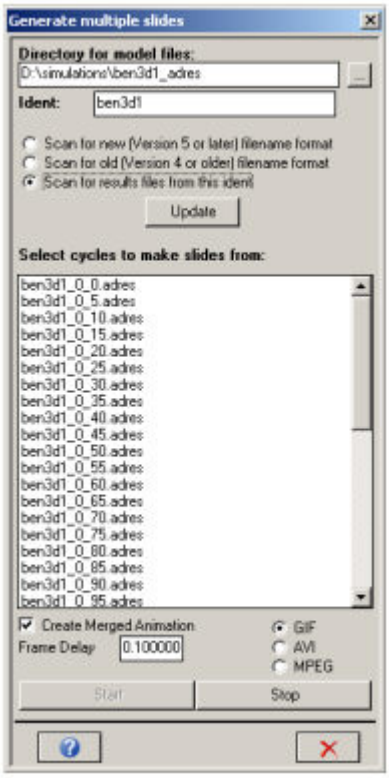
For information on the **Slides Setup** button see Pull-Down Menu - View - Slideshow - Setup (p. 82).

6.1.2. Capture Current Image

Use the **Capture Current Image** button to capture a slide from the current contents of the graphics window. The slide will be generated in the slides directory, which is defined using the Slides Setup (p. 82) button.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

6.1.3. Capture Sequence



This window lets you generate images from the restart or result files of a completed analysis. The output of the operation may either be separate images or a merged animation.

Directory for Model Files

Choose a directory to scan for file.

Ident

Choose the file ident (model name) to scan for.

Scan for Current Filename Format

Files of the format *ident*_cycle.ad will be displayed in the selection panel.

Scan for Old (Verson 4 or Older) Filename Format

Files of the format *ident*.cycle will be displayed in the selection panel.

Update

Update the list of cycles displayed in the selection panel, after any of the display criteria have been changed.

Select Cycles to Make Slides From

Choose which restart or results files shown in the list are to be used to create images.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Create Merged Animation

Select this option to create a single file as an animation. Select the format of animation created (GIF, AVI or MPEG). If this option is not selected, the animation will be created as a series of separate images.

Frame

Specify the delay per frame (in seconds) for merged animations.

Start

Begin the image generation operation.

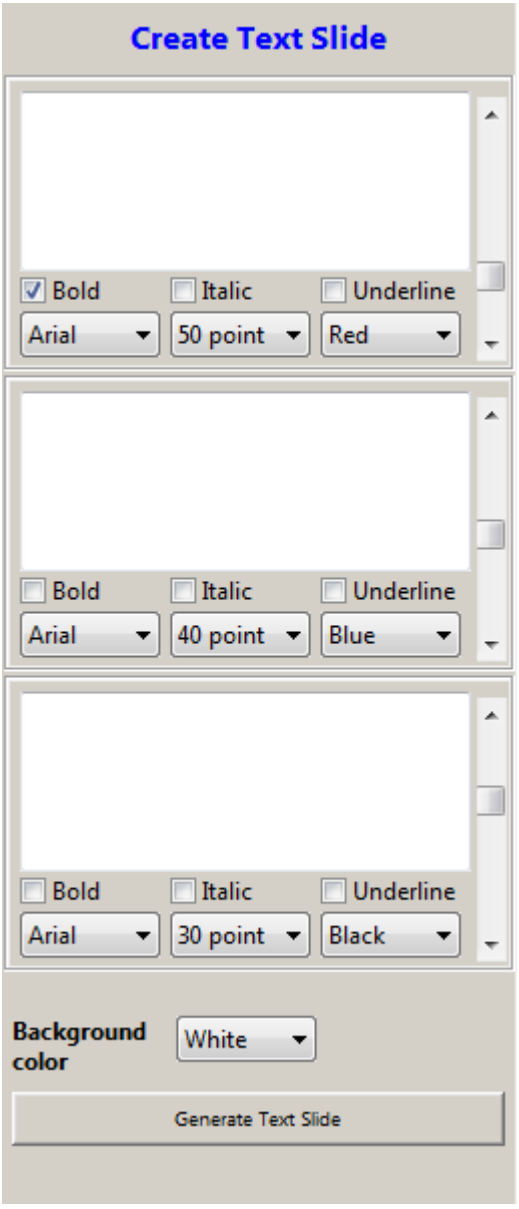
Stop

Interrupt the image generation operation if it is currently in progress.

6.1.4. Text Slide

Use the **Create Text Slide** feature to generate text-only slides which can be inserted into an animation.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



The **Create Text Slide** panel consists of three identical sections. Enter text and use the vertical scroll bars to position the text in the graphics window. Options are available to select the font type, size, and color, and also to make the font bold, italic, or underlined.

Background Color

Select the background color for the text slide (black or white).

Generate Text Slide

Create the text slide once the layout is complete. The slide will be created in the slides directory, which is defined using the Slides Setup (p. 82) button.

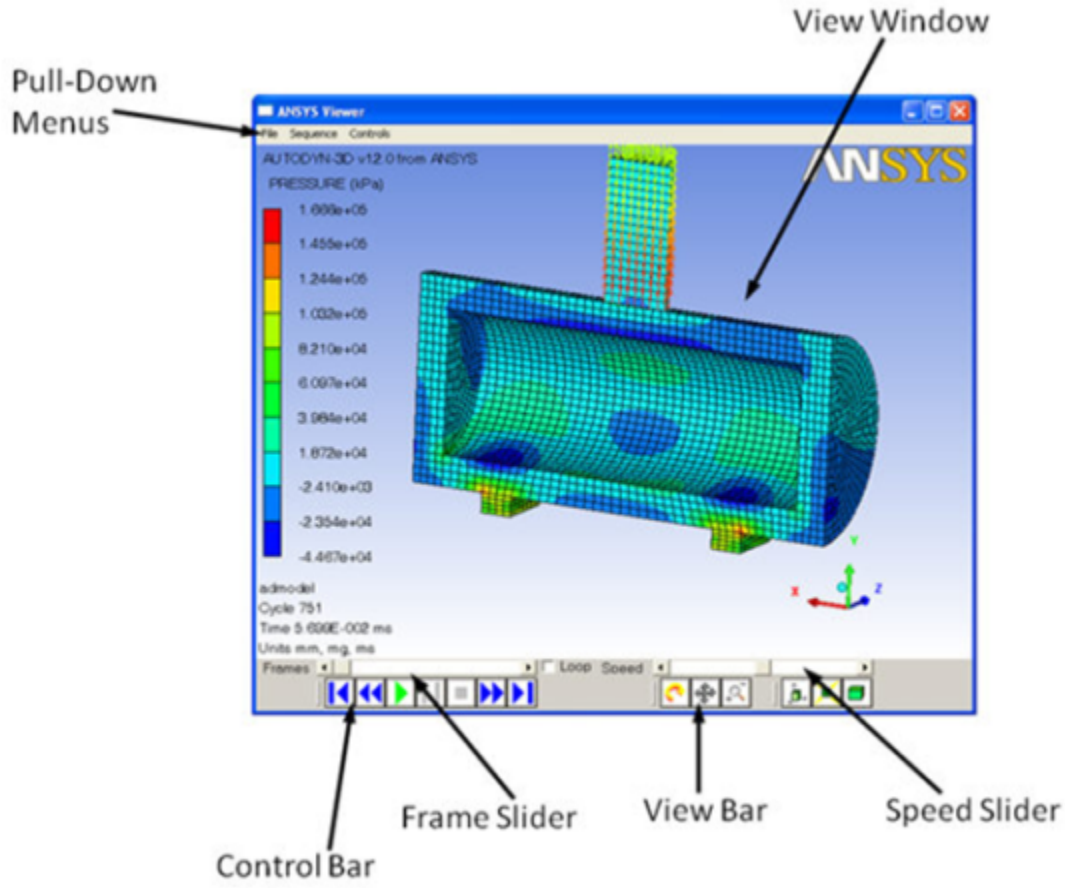
Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

6.2. View Slides

The View Slides option on the Navigation Bar starts the Slide Viewer (it can also be run independently of Autodyn).

The Viewer lets you load and view GIF and GFA animations.

Figure 6.1: Components of the Viewer's user interface

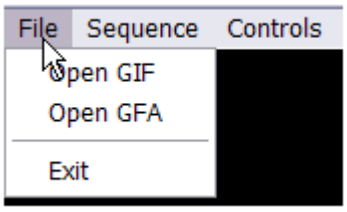


View Window

This is where your animations are shown.

Pull-down Menus

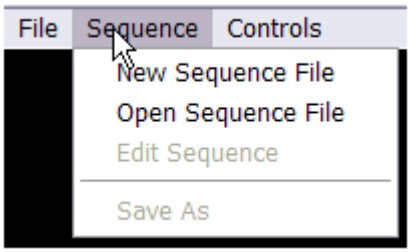
- **File**



Enables you to load single GIF and GFA animation files for playback, and to exit the program.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- **Sequence**



Enables you to playback multiple GIF animations using sequence files.

A sequence file contains a list of GIF animations you want to view consecutively.

All the animations in a sequence file must be in the same directory as the sequence file.

– **New Sequence File**

Opens a new sequence file.

– **Open Sequence File**

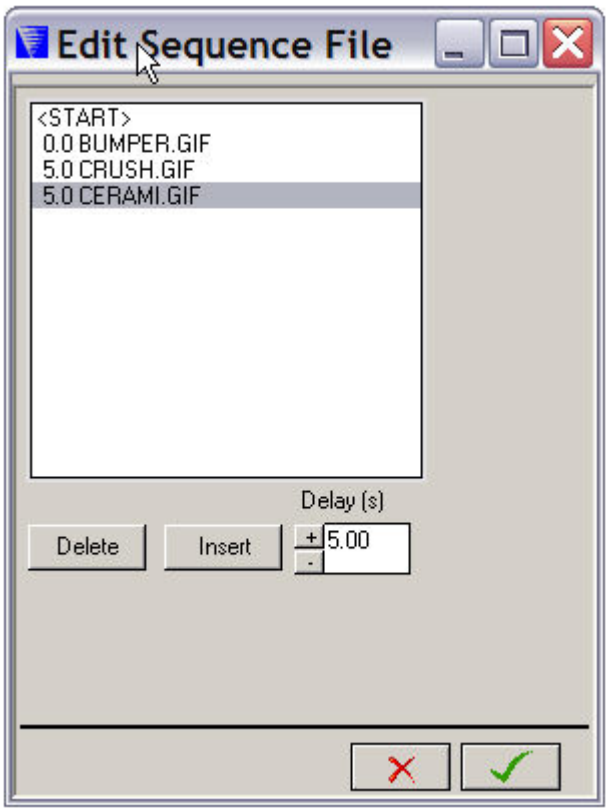
Opens an existing sequence file.

– **Edit Sequence**

Enables you to insert and delete animations in the current sequence file.

The following dialog window appears when you select this option:

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



The box at the top of this dialog contains a list of the animations currently included in the sequence, together with their time delay (how long the first slide in the sequence is shown, in seconds).

→ **Insert**

Inserts another animation immediately after the selected animation.

→ **Delete**

Deletes the selected animation.

→ **Delay**

Enter a time delay in this field (in seconds). When you Insert an animation, this time delay is applied.

You can create and edit sequence files manually if you wish. Sequence files must have a `.seq` extension and use the following format:

Format	**Example**
The number of animations in the sequence	3
Default frame delay in seconds + <space> + file name	0.0 anim.gif
Default frame delay in seconds + <space> + file name	1.2 another.gif

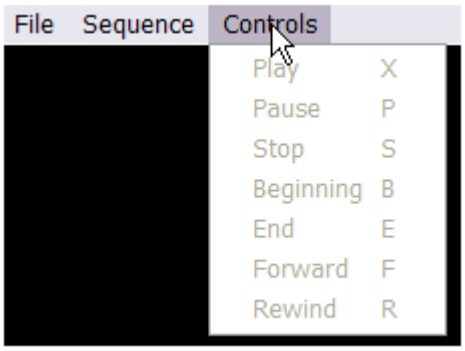
Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Format	Example
Default frame delay in seconds + <space> + file name	1.6 yetanother.gif

– **Save As**

Saves the current sequence file (`.seq`)

- **Controls**



Provides the same functions that are available in the Control Bar.

Control Bar

Controls the playing of the GIF or GFA animation you have loaded.

Clicking a button on this bar performs the following actions (you can type the corresponding **Hotkey** as an alternative to pressing a button).

Button	Hotkey	Action
	b	Go to the first frame of the animation
	r	Go back one frame
	x	Play (from the current frame)
	p	Pause (at the current frame). This is a toggle button.
	s	Stop (at the current frame)
	a	Advance one frame
	e	Go to the last frame of the animation

View Bar

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

This bar appears only when you have loaded a GFA animation.

The buttons on this bar let you control how you view your GFA animation and work in the same way as the corresponding icons in the Autodyn **toolbar**.

Rotate view
Translate view
Zoom in or out
Set view (brings up a window that allows you to set an exact view)
Reset view (to the initial view)
Turn perspective on or off (toggle)

Frames Slider

Enables you to move quickly to any frame in the animation(s).

Speed Slider

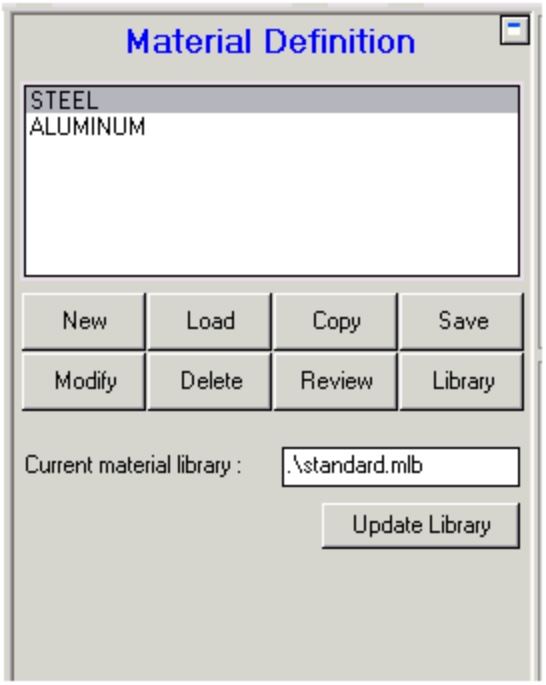
Adjusts the playback speed for animations.

Loop

Check this box if you want the animation(s) to automatically start again from the first frame when the last frame is reached.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Chapter 7: Autodyn Materials



This panel lets you define the materials you want to use in your model.

Material List

Lists the materials you have currently defined for your model. You can select materials in this list.

New

Define a new material.

Modify

Modify parameters for the selected material.

Copy

Copy material component data from an existing material to either a new material or another existing material.

Delete

Delete one or more materials from your model.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Review

Review the parameters for the selected material in a browser window.

Library

The default material library is **standard.mlb**. This contains all the material data provided by Ansys, Inc.

Click **Library** if you want to change the material library to one you have obtained or created.

Note:

The user-defined library should always be named `standard` as the code defaults to this library name once you leave the Materials panel. Alternatively, you can name the library differently but will have to reload it every time you return to the Materials panel.

Load

Load materials from your current material library.

Save

Save the materials you have defined to a material library.

Current material library

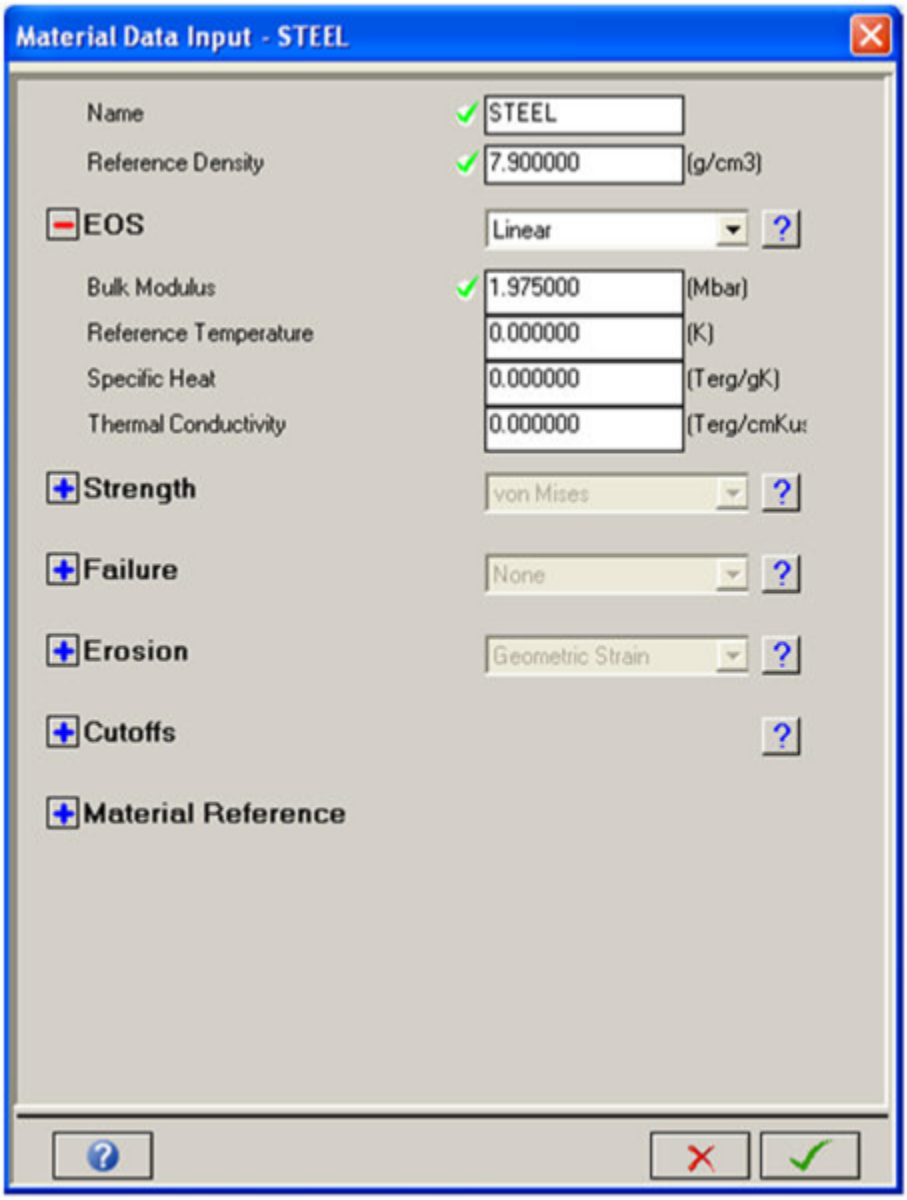
The current material library you have selected.

Update Library

Update old material library files, so that they can be used with this version of Autodyn.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

7.1. Materials - New/Modify



This window lets you define or modify the model components and parameters used for a material.

Name

The name for your material.

Reference Density

The reference density (usually the initial density) for your material.

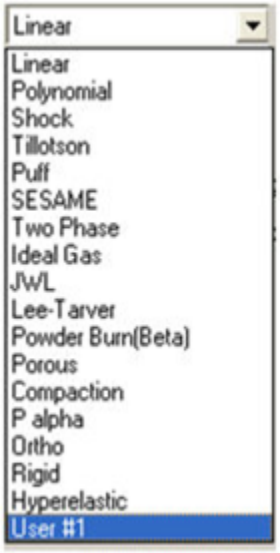
EOS

Define the Equation of State you want to use for your material.

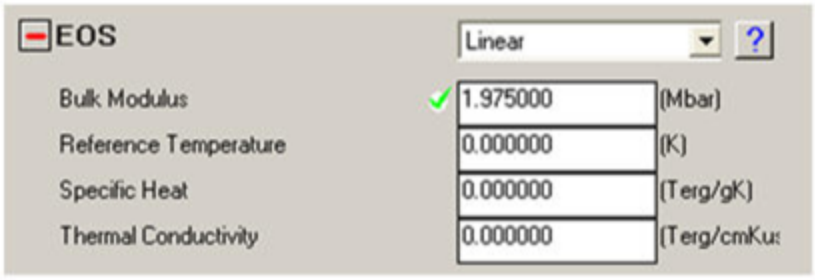
Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Click + next to this option to expand the dialog.

Select the Equation of State you want to use from the pull-down menu :



Then type in parameter values in the fields provided :



Strength

Define the strength model you want to use for your material.

Click + next to this option to expand the dialog.

Select the strength model you want to use from the pull-down menu:

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

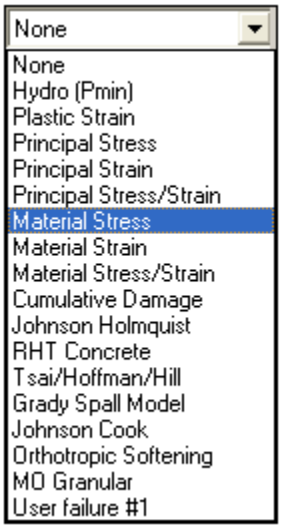
Then type in parameter values in the fields provided :

Failure

Define the failure model you want to use for your material.

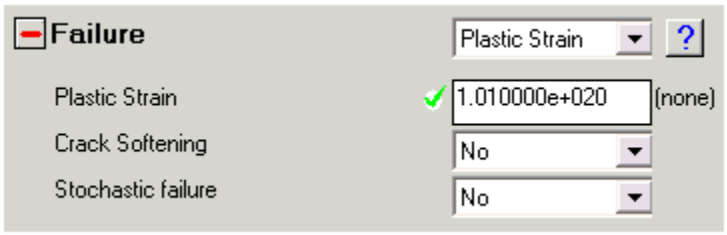
Click + next to this option to expand the dialog.

Select the failure model you want to use from the pull-down menu:



Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

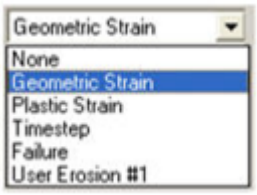
Then type in parameter values in the fields provided:



Erosion

Define the erosion model you want to use for your material.

1. Click + next to this option to expand the dialog.
2. Select the erosion model you want to use from the pull-down menu:



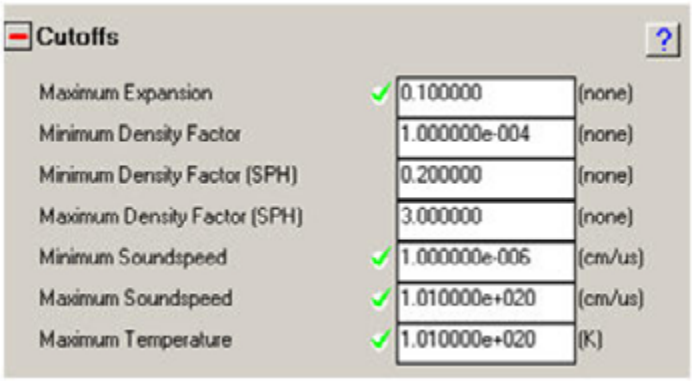
3. Type in parameter values in the fields provided:

Cutoffs

Change the cutoffs for your material.

It is best to use the default values for these cutoff if you are using the default units (mm, mg, ms).

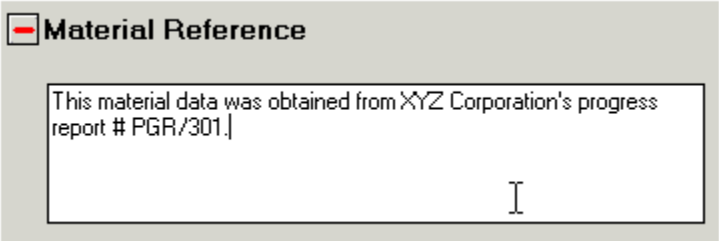
Click + next to this option to expand the dialog and change whatever cutoffs you want.



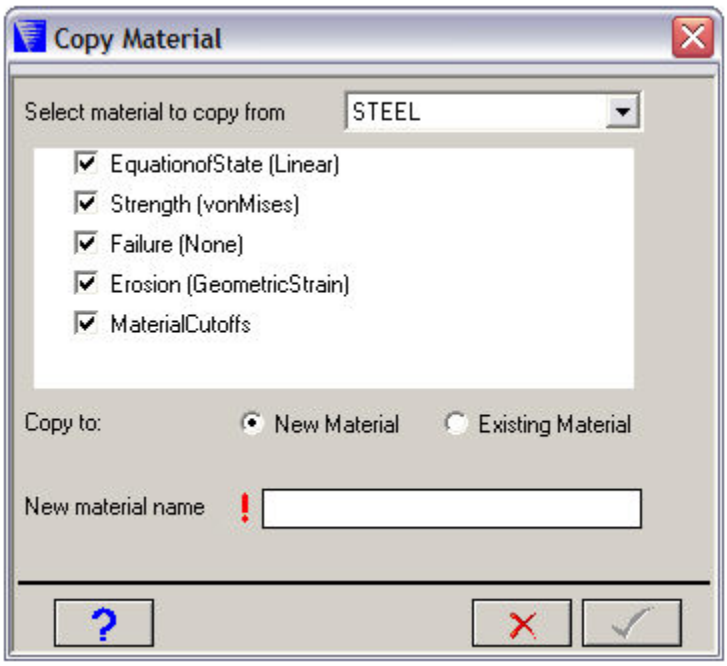
Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Material Reference

Click + next to this option to expand the dialog and type in a reference for your material in the field provided.



7.2. Materials - Copy



This window lets you copy component data from an existing material to either a new material or another existing material.

Select material to copy from

The material you want to copy.

Select data components to copy

Check the boxes beside the components you want to include in the copy process.

Copy to

Copy the data to a new material or an existing material:

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- If you copy to a new material, components not copied will be set to the default state.
- If you copy to an existing material, components not copied will retain their existing state.

New material name

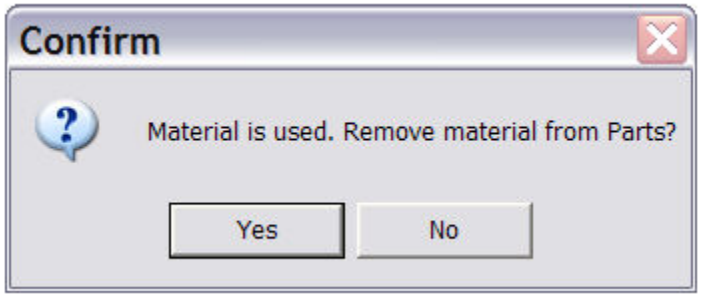
If you choose to copy to a new material, enter your name for the new material.

7.3. Materials - Delete

This window lets you delete materials from your model.

Select the material(s) you want to delete and click .

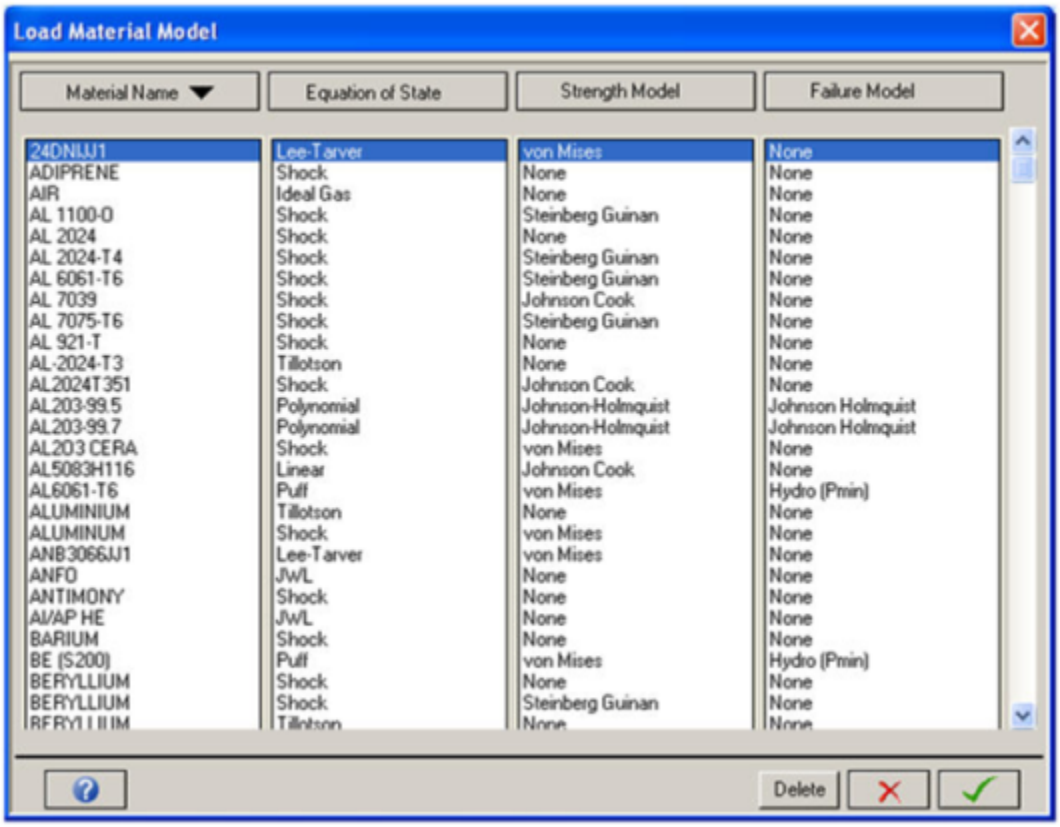
If the material you want to delete has been assigned to one or more Parts, you will be asked to confirm that it is OK to remove the material from the Part(s):



Elements that have their material removed become **Unused**.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

7.4. Materials - Load



This window lets you load materials from the current Material Library.

The available materials are listed in a vertical scroll box. For each material four columns provide its **Material Name**, **Equation of State**, **Strength Model**, and **Failure Model**.

Click the button at the top of each column to sort materials alphanumerically in that column.

Using the **Ctrl** and **Shift** keys as necessary, you can select one or more materials.

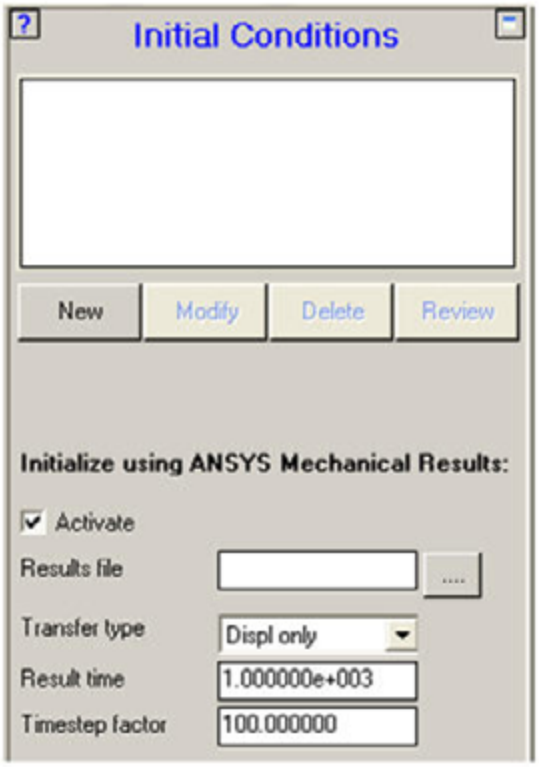
To load the selected materials into your model, click .

To delete the selected materials from the library, click .

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information
of ANSYS, Inc. and its subsidiaries and affiliates.

Chapter 8: Autodyn Initial Conditions



This panel lets you create **Initial Condition Sets** that can be used in **Fill** operations for **Parts**.

You do not have to use an Initial Condition Set to fill a Part, but there is a major advantage in using one.

Any subsequent changes you make to an Initial Condition Set are automatically applied to the Parts that have been filled using that Initial Condition Set.

This means you do not have to refill the Parts (this is currently not true for Euler Parts).

Initial Condition Set List

The box at the top of this panel lists the Initial Condition Sets you have currently defined for your model. You can select an Initial Condition Set in this list.

New

Define a new Initial Condition Set.

Modify

Modify the selected Initial Condition Set.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Delete

Delete one or more Initial Condition Sets from your model.

Review

Review the parameters for the selected Initial Condition Set in a browser window.

Implicit-Explicit Model Initialization

Activate

Initialize an Autodyn analysis model using the results of an Ansys Mechanical implicit analysis.

Results File

The Ansys `.rst` results file to be used to initialize the Autodyn analyses.

Transfer Type

The initialization method; either the Displacement Only or the Material State option as described below.

Results Time

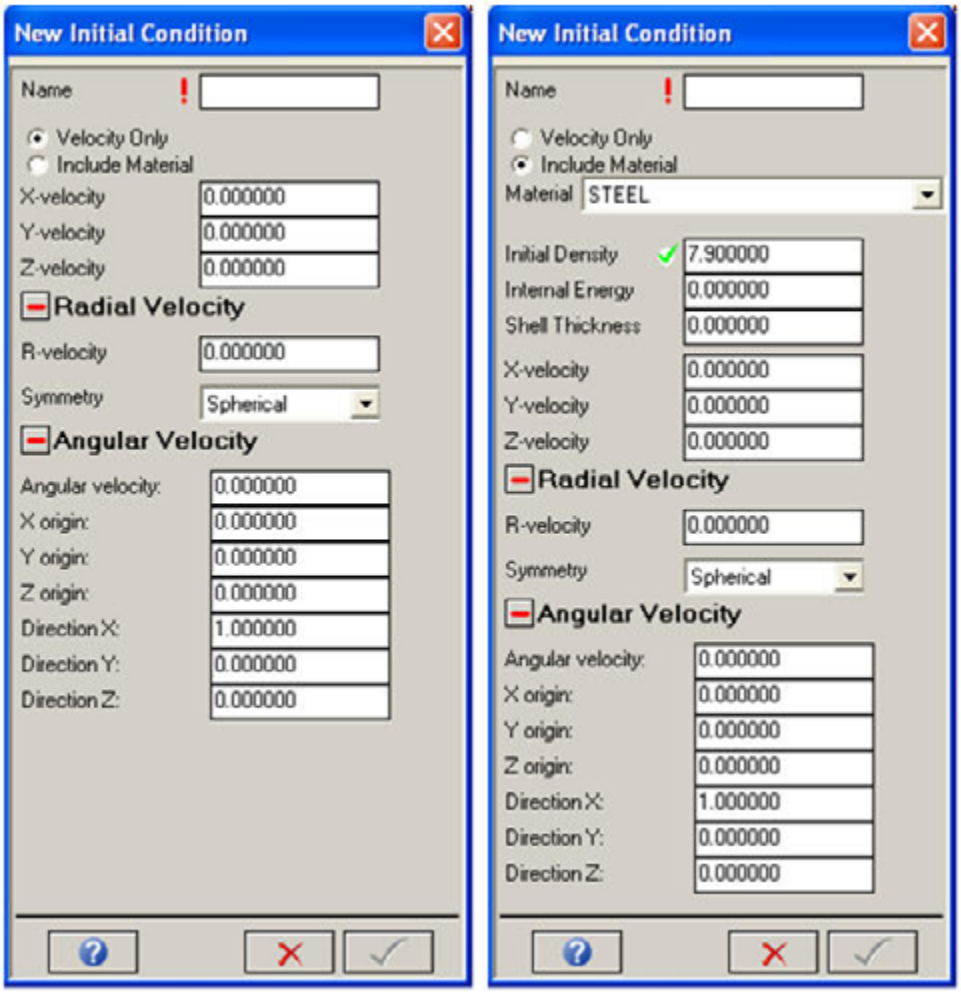
The result time from the Ansys Mechanical implicit analysis at which to extract results from the `.rst` file.

Timestep factor

(Note that this option is applicable only for the Displacement Only method.) The initial timestep from the Autodyn solution is multiplied by the timestep factor. The resulting time (PVELTM) is used together with the nodal displacements from the Ansys Mechanical analysis to calculate constant nodal velocities. These nodal velocities are applied to the Autodyn model over PVELTM in order to initialize the Autodyn nodes to the correct positions.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

8.1. Initial Conditions - New/Modify



This window lets you define a new Initial Condition Set.

Name

The Initial Condition Set.

Velocity Only

Select this option if you want your Initial Condition Set to only include velocities (top left window).

Include Material

Select this option if you want your Initial Condition Set to include velocities and a material and its associated variables (top right window)

Material

The material you want to assign to the Initial Condition Set.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Material variables

The material density and internal energy to be used with this initial condition.

In addition, if the initial condition is to be used with a shell element, the element thickness can be specified. Note that this must be used for the unstructured solvers.

Translational Velocity

The global X, Y, and Z velocity associated with the initial condition.

Radial Velocity

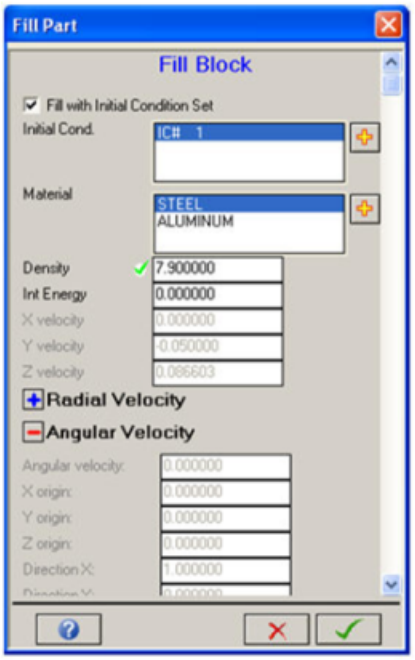
An initial velocity of fixed magnitude. The direction of the velocity will be defined by the radial position of each node from the original (0.,0.,0.).

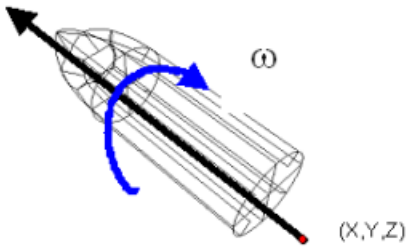
Spherical and cylindrical symmetry options are available.

Angular Velocity (3D only)

An angular velocity can be specified in addition to translational velocity in Autodyn-3D.

The angular velocity is defined in radians / time and the axis of rotation is defined through the direction cosines of a vector passing through a point in space.





During the Fill of a Part, the angular velocity and spin axis are used to calculate the resultant X, Y, and Z velocity at each node. These are then added to any specified translational velocities.

8.2. Initial Conditions - Explicit Model Initialization

Autodyn explicit analysis can be initialized (or pre-stressed) with the results from either a linear static structural, non-linear static structural or transient dynamic Ansys Mechanical implicit analysis. Note that this feature requires the Ansys results to be saved to a .rst file.

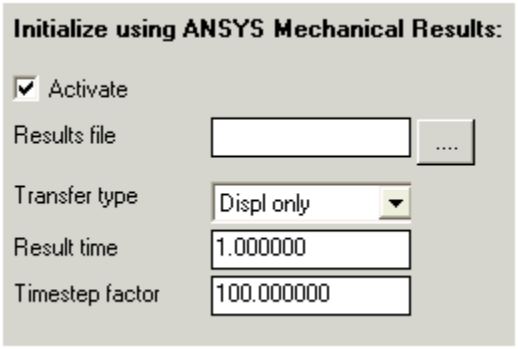
Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

It is assumed that the same mesh is always used in both the implicit and explicit analyses; otherwise results will not be mapped onto the explicit mesh correctly. Note that quadratic elements may be used in the implicit model. In this case results for the corner nodes and average results for the element will be applied to the explicit mesh.

You are advised to check the `autodyn.log` file shortly after starting the Autodyn analysis as any errors occurring during the implicit results transfer will be echoed to this file.

Activate

Initialization of an explicit analysis under implicit analysis results can be selected using the **Activate** option under **Initialize using Ansys Mechanical Results**.



The menu shown above is used to configure the transfer of data from the Ansys Mechanical to the Autodyn solution and in which manner Autodyn will be initialized.

Results File

Select the Ansys `.rst` results file to be used to initialize the Autodyn analysis.

Transfer Type

Either the **Displacement Only** or the **Material State** option as described below.

Transfer Type options are available to define what analysis results need to be transferred from the Ansys implicit analysis to initialize the Autodyn analysis:

1. **Displacement Only**

   Nodal displacements from an Ansys linear static solution are used to initialize the Autodyn nodal positions. These displacements are converted to constant nodal velocities and applied for a pre-defined time in order to obtain the required displaced co-ordinates. During this time elements stress/strains are calculated as normal by Autodyn. Once the displaced nodal positions are achieved all nodal velocities are set to 0 and the solution is completely initialized.

   - This option is applicable to unstructured solids (hexahedral and tetrahedral), shells and beams.
   - This option is only applicable to results from a linear, static structural analysis.
   - Linear elastic materials only can be used in the explicit analysis when using the Displacement Only Option.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- Note that after Autodyn model initialization static damping (**Controls** > **Damping Options** > **Static Damping**) needs to be applied to change the Autodyn calculation from a dynamic solution to a state of stress equilibrium. For optimal convergence, the value chosen for the damping constant, R, may be defined by:

```
R=2*timestep/T
```

  where timestep is the expected average value of the timestep and T is the longest period of vibration for the system being analyzed.

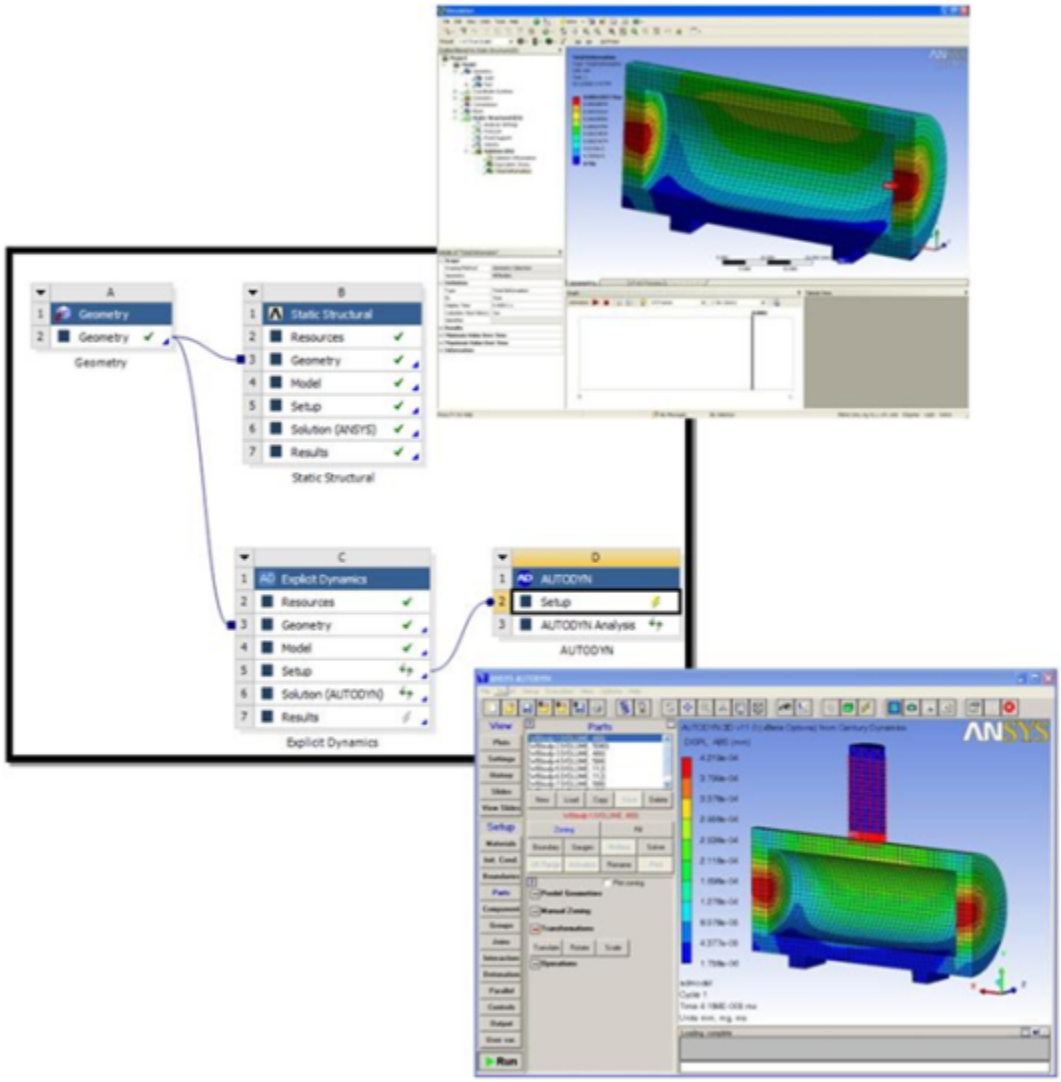
2. **Material State**

   Nodal displacements and element stresses, strains, plastic strains, velocities, and temperatures from an Ansys (thermal) solution are used to initialize an Autodyn analysis at cycle 0.

   The Autodyn solution is therefore completely initialized at the start of the analysis.

   - This option is only applicable to unstructured solid elements (hexahedra and tetrahedral).
   - This option is applicable to results from a linear static structural, nonlinear static structural, or transient dynamic Mechanical system. The Ansys solution may be preceded with a Steady-State Thermal solution in order to introduce Temperature differences into the solution. Note that the accompanying thermal stresses due to the thermal expansion coefficient will be transferred but may dissipate since the thermal expansion coefficient is not taken into account in the Explicit Dynamics analysis.
   - The following materials can be used in the Autodyn solution with the Material State Option. Equations of State (linear, hyperelastic). Strength Models (elastic, von-Mises Johnson-Cook, piecewise, Zerilli Armstrong, Steinberg Guinan, Cowper Symonds, hyperelastic, bilinear isotropic hardening).
   - Note that static damping can also be applied by using a static damping constant, but this may not be required.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Results Time

The result time from the Ansys Mechanical implicit analysis at which to extract results from the `.rst` file. For a linear static solution this will always be 1.0. For a nonlinear or transient dynamic solution this time will correspond to a result time from the Ansys Mechanical analysis.

Timestep factor

Note:

This option is applicable only for the Displacement Only method.

The initial timestep from the Autodyn solution is multiplied by the timestep factor. The resulting time (PVELTM) is used together with the nodal displacements from the Ansys Mechanical analysis to calculate constant nodal velocities. These nodal velocities are applied to the Autodyn model over PVELTM in order to initialize the Autodyn nodes to the correct positions.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

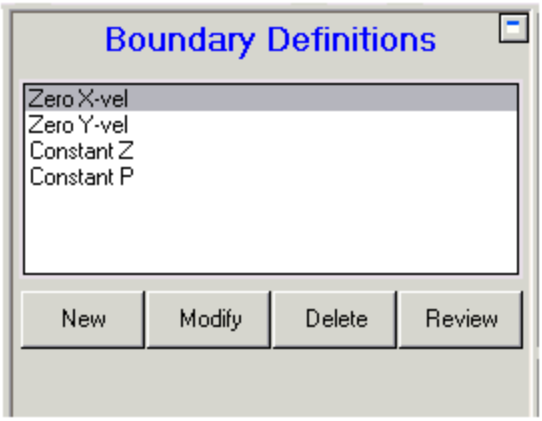
8.3. Initial Conditions - Delete

This window lets you delete Initial Condition Sets from your model.

Select the Initial Condition Set(s) you want to delete and click .

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Chapter 9: Autodyn Boundaries



This panel lets you create **Boundary Conditions** that can be applied to Parts.

Boundary Condition List

The box at the top of this panel lists the Boundary Conditions you have currently defined for your model. You can select a Boundary Condition in this list.

New

Defines a new Boundary Condition.

Modify

Modifies the selected Boundary Condition.

Delete

Deletes one or more Boundary Conditions from your model.

Review

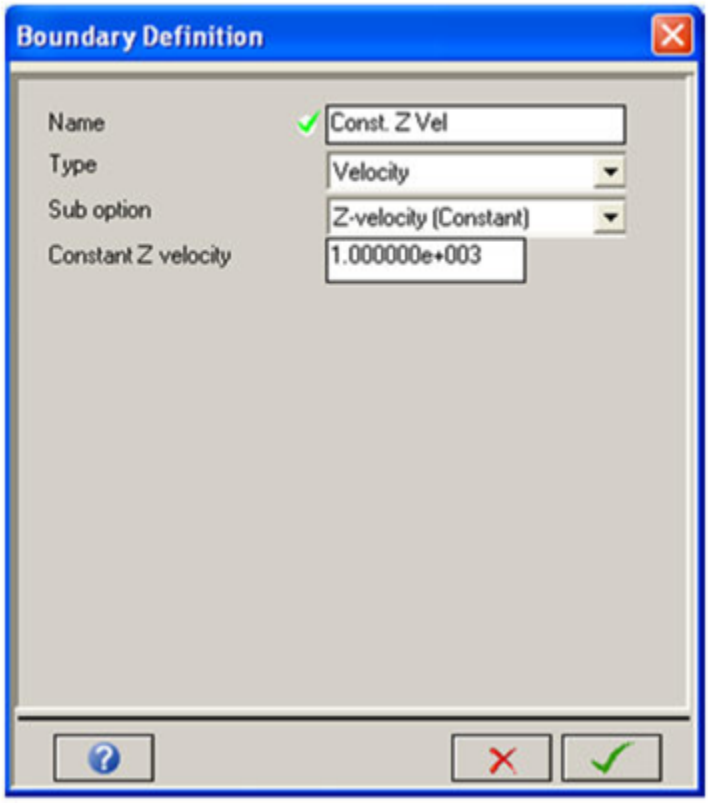
Reviews the parameters for all Boundary Conditions in a browser window.

Note:

General Velocity and General Force boundary conditions that are defined in the Workbench Mechanical application interface cannot be modified in the Autodyn application. They can, however, be visualized; see Settings - Boundary (p. 118).

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

9.1. Boundaries - New/Modify



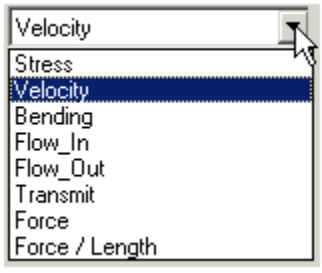
This window lets you define a new boundary condition.

Name

The name for the boundary condition.

Type

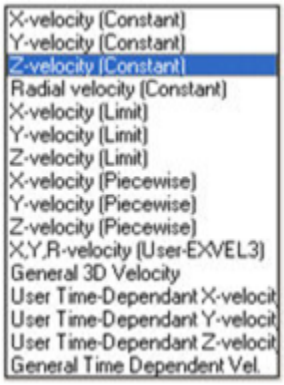
The type of boundary condition you want to use :



Sub Option

The sub-option you want to use :

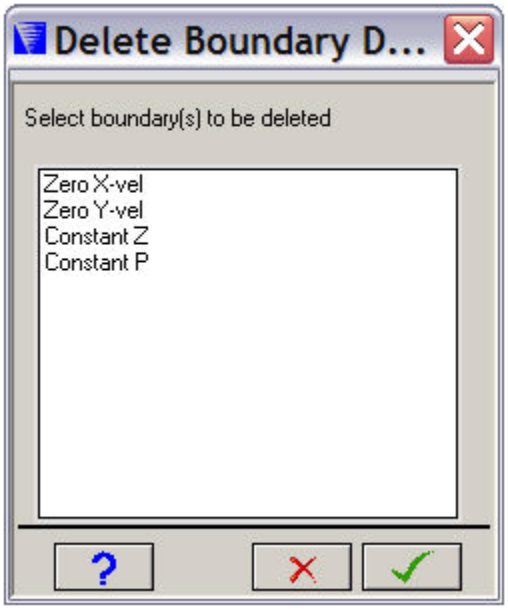
Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Parameters

The values for any parameters associated with your selection (e.g. Constant Z Velocity).

9.2. Boundaries - Delete



This window lets you delete Boundary Conditions from your model.

Select the Boundary Condition(s) you want to delete and click .

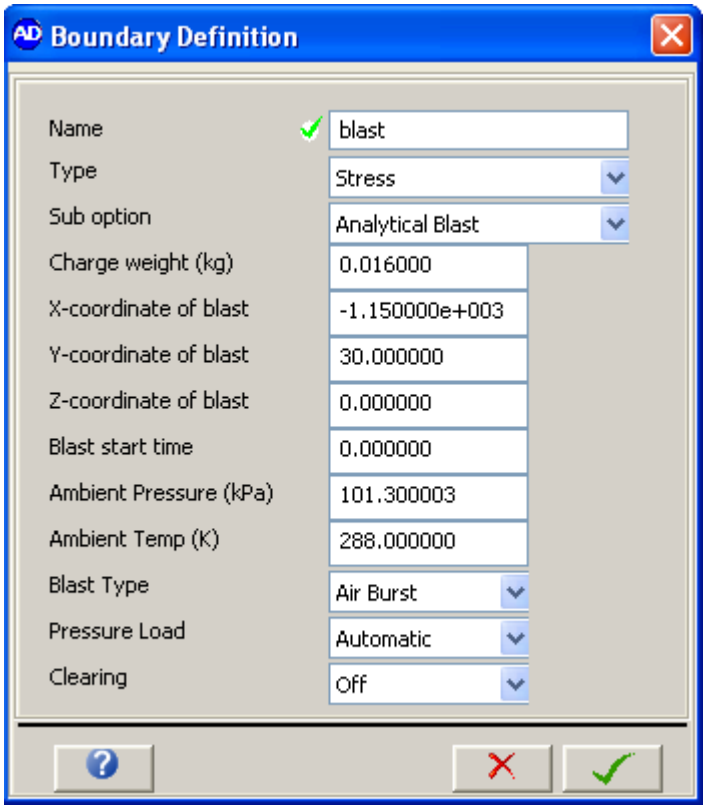
9.3. Analytical Blast Boundary Condition

The analytical blast boundary condition can be used to apply pressure loads to faces on structured and unstructured volume and shell elements due to the detonation of high explosives. The implementation uses an interpretation of the methods presented in United States Army Technical Manual, TM 5-855-1 [1].

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

9.3.1. User Interface

An Analytical Blast boundary condition can be defined and applied to parts in the same manner in which a stress boundary would be. The blast boundary condition parameters that need to be specified are as shown below.



The numerical inputs are as follows:

Parameter	Description	Notes
Charge weight (kg)	Weight of charge in equivalent weight of TNT (in kg)	Must be greater than 0.
X-coordinate of blast	X-coordinate of blast center in model units	
Y-coordinate of blast	Y-coordinate of blast center in model units	
Z-coordinate of blast (3D only)	Z-coordinate of blast center in model units	
Blast start time	Detonation time in model time units	
Ambient Pressure (kPa)	Ambient pressure in kPa	Must be greater than or equal to 0.
Ambient Temp (K)	Ambient temperature in Kelvin	Must be greater than or equal to 0.

Additionally, the following options can be specified:

- **Blast Type**: options are Air Burst or Surface Burst.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- **Pressure Load**: options are Automatic, Always Facing, or Never Facing. If Automatic is chosen, the solver determines whether the surface faces the blast in the following manner: if the face normal points away from the blast, then the surface faces the blast (NB for volumes the face normal is assumed to point into the volume). Therefore when applying a blast boundary condition to shells it is necessary to ensure that the shell normals are oriented as required. If Always Facing or Never Facing are chosen, the surfaces to which the blast boundary condition is applied are assumed to be always or never facing the blast, respectively, irrespective of the direction of the face normals. Faces determined to be facing the blast receive the reflected pressure load, while those that are determined not to face the blast receive the incident side on overpressure. The default is Automatic.
- **Clearing**: can be set to be On or Off. The default is Off. Please read the information regarding clearing in Limitations (p. 185).

9.3.2. Theory

The following Analytical Blast topics are discussed in this Theory section:

9.3.2.1. Pressure Load Evaluation Without Clearing
9.3.2.2. Clearing 'On'
9.3.2.3. BLAST.LOAD Variable
9.3.2.4. Limitations
9.3.2.5. Example – Small Scale Urban Blast

9.3.2.1. Pressure Load Evaluation Without Clearing

When clearing is switched off, the impulse on a face which faces the blast is determined without account for pressure relief from structure edges. The applied pressure load on a face depends on the distance from the blast, the angle of incidence of the blast wave, and the orientation of the face with respect to the blast. The angle of incidence, θ, is computed by finding the angle between the face normal and the vector joining the face center and the blast center. The distance from the blast, d, is computed as the straight-line distance between the blast center and the face center.

The scaled distance, $z=d/\sqrt[3]{W}$,is computed from the blast weight, W, and is used to compute the blast parameters including the incident overpressure, $P_O{}^I$, positive phase duration, t_0 , arrival time, t_{arr} , and incident impulse, I_I , using the formulae reported in 1984 report ARBRL-TR-02555 by Kingery and Bulmash [2].

If the face does not face the blast, the applied pressure load is given by the Friedlander equation [3] for side on blast loads:

$$P(t)=P_O^I\left(1-\frac{t-t_{\text{arr}}}{t_0}\right)\exp\left(-\lambda_I\frac{t-t_{\text{arr}}}{t_0}\right) \tag{9.1}$$

where the decay parameter, λ_I , is calculated such that $\int_{t_{arr}}^{t_{arr}+t_0}P(t)\,dt=I_I$. If t is less than t_{arr} then no pressure is applied.

However, if the face is determined to face the blast, the normal reflected impulse (for 0° angle of incidence), I_R , is also computed using the formulae reported in Kingery and Bulmash [2] and the peak reflected over pressure, $P_O{}^R$, is computed from the angle of incidence and $P_O{}^I$ using Figure 3-3 in TM 5-855-1 [1]. The reflected impulse is corrected for non-normal angles of incidence using

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

$$I=I_I\left(1+\cos\theta-2\cos^2\theta\right)+I_R\cos^2\theta \tag{9.2}$$

and the applied pressure load is given by the Friedlander equation

$$P(t)=P_O^R\left(1-\frac{t-t_{\text{arr}}}{t_0}\right)\exp\left(-\lambda_R\frac{t-t_{\text{arr}}}{t_0}\right) \tag{9.3}$$

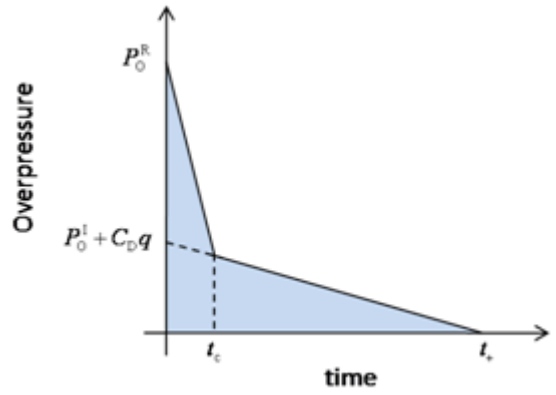
where λ_R is calculated such that $\int_{t_{arr}}^{t_{arr}+t_0} P(t)\,dt=I$.

9.3.2.2. Clearing 'On'

When clearing is switched on, the impulse on a face which faces the blast is corrected to account for pressure relief from structure edges as described in TM 5-855-1 [1].

The time for the reflected pressure wave to be cleared, t_c , is computed as $t_c = 3l/u$ where l is the distance to the nearest edge from which pressure relief can occur, and u is the blast wave speed (calculated using the equations in Kingery and Bulmash [2]). A fictitious pressure-time curve as shown in Figure 9.1: Overpressure-Time Curve (p. 184) is constructed which is used to compute the new impulse, I_c (area shaded in Figure 9.1: Overpressure-Time Curve (p. 184)). This approach can overestimate the impulse (typically when the pressure loads are large), so the impulse is capped by the reflected impulse as given in Equation 9.2 (p. 184).

Figure 9.1: Overpressure-Time Curve



Overpressure-time curve used to calculate the reduced impulse on a structure due to clearing. q is the dynamic pressure which is computed from Figure 3-8 in TM 5-855-1 [1] using P_O^I and θ. C_D is the drag coefficient which is assumed to be 1 [1] and $t_+=2I_I/P_O^I$.

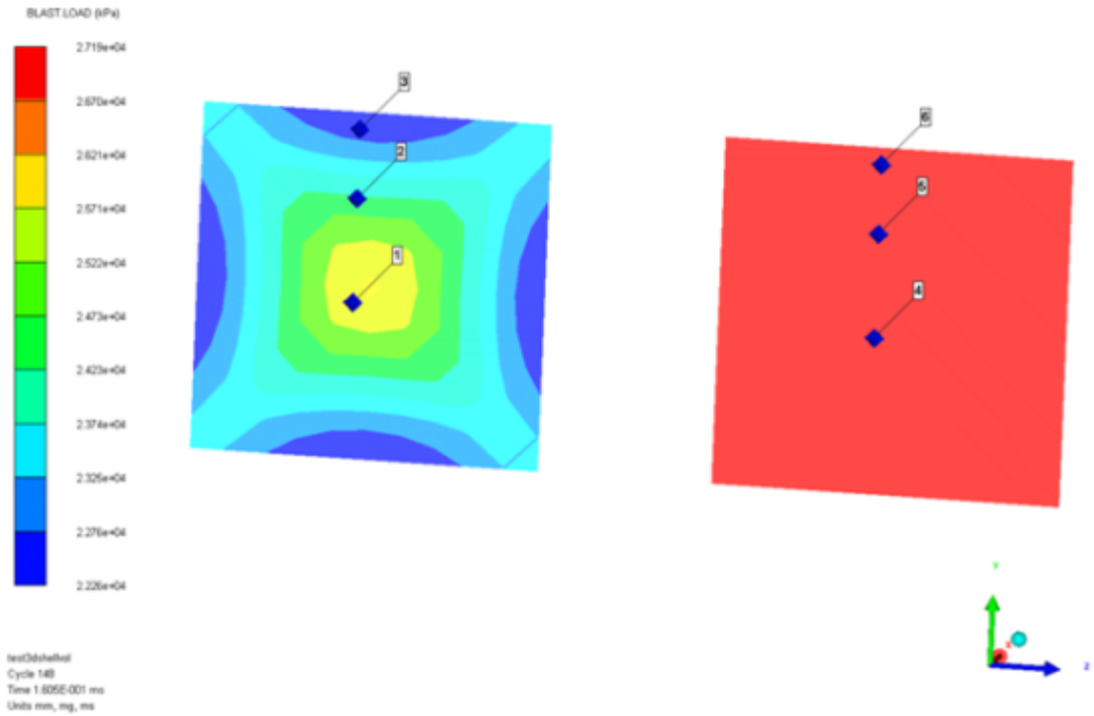
The pressure-load applied to a face is a Friedlander equation

$$P(t)=P_O^R\left(1-\frac{t-t_{\text{arr}}}{t_0}\right)\exp\left(-\lambda_c\frac{t-t_{\text{arr}}}{t_0}\right) \tag{9.4}$$

where the decay parameter, λ_c , is taken such that $\int_{t_{arr}}^{t_{arr}+t_0} P(t)\,dt=I_c$. Figure 9.2: Effect of the Clearing Option on Pressure Load (p. 185) shows the difference in pressure load applied to two shell parts both with analytical blast boundary conditions applied, but one with clearing on and the other with clearing off.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 9.2: Effect of the Clearing Option on Pressure Load



The effect of the clearing option on pressure load applied due to an analytical blast boundary condition. Both shells have analytical blast boundary conditions applied with the center of the charge in-line with the center of the shell and positioned 500 mm away (charge weight is 1 kg). The shell on the left has clearing turned on and the shell on the right has clearing turned off. The snapshot has been taken approximately 6 µs after the blast arrival at the shells.

9.3.2.3. BLAST.LOAD Variable

The load applied to a face due to an analytical blast boundary condition can be visualised both using contour plots and history plots at gauges using the variable BLAST.LOAD. This variable is only available for contour plots when an analytical blast boundary condition has been applied to some parts. The pressure load applied to a face due to an analytical blast boundary condition is plotted in the element to which the face belongs. Note that if two faces of an element both have analytical blast boundary conditions applied (e.g. at corners), from which face the value of BLAST.LOAD is taken is arbitrary. In order to plot the BLAST.LOAD variable in a history plot, it needs activating in the Select Gauge Variables (under History in the Outputs tab) menu prior to solving.

9.3.2.4. Limitations

The formulae in ARBRL-TR-02555 for the blast parameters are only valid for scaled distances (ratio of distance to blast and the cube root of the blast weight) between 0.147 and 40.0 m/kg$^{1/3}$, therefore, it is necessary to ensure that all faces with an analytical blast boundary condition applied satisfy this condition (if the scaled distance between the face and blast is not within this range the pressure load is set to zero).

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

The blast parameters are computed only once for each face during the model initialization, which therefore assumes negligible movement of the faces to which the analytical blast boundary condition are applied during the duration of the blast loading.

This implementation does not take into account shadowing, diffraction, multiple reflections or channelling by structures in the model as the distance of a face from the blast is computed as the straight line distance between the blast and the face. This is also true for faces not facing the blast which can lead to overestimation of pressure loads and impulses on these faces. It is assumed that the face has a direct line of sight to the blast if the face is determined to face the blast origin, irrespective of any structured obstructing the direct line of sight. Furthermore, since reflections from other structures are not considered, the implementation is not applicable for internal blasts or for buried charges.

The clearing option is intended only for very simple geometries and is designed only to work if each analytical blast boundary condition is applied only to one part. 'Edges' from which clearing can occur for a particular blast boundary condition on a particular part are determined by finding the blast load faces that face the blast which share a common edge only with faces that either have no analytical blast boundary condition applied, or have blast boundary conditions applied but do not face the blast. The clearing distance from a face is then calculated by finding the distance to the nearest 'edge' face. The distance is taken as the straight-line distance between the center of the two faces.

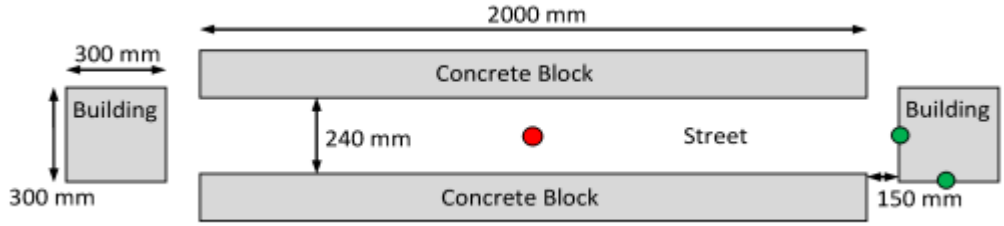
Joins are not accounted for when determining the 'edge' faces, therefore clearing could potentially come from the edge of joined faces.

The solver has no concept of the plane in which the floor lies in for surface bursts. Therefore clearing could potentially come from what the user intends to be the floor. As clearing cannot come from symmetry planes, a symmetry plane could be used to represent the floor.

9.3.2.5. Example – Small Scale Urban Blast

The results of an Autodyn analytical blast simulation were compared to the results of a 1/50th scale experimental study of a blast in an urban environment as described by Feng [4]. The experiments consisted of an 8 g TNT charge positioned 30 mm above a horizontal steel plate. Structures representing buildings on the street were rested on the plate with the charge at the street center as depicted in Figure 9.3: Small Scale Blast Experiment (p. 186). Two experiments were conducted: one with and one without the concrete blocks shown in Figure 9.3: Small Scale Blast Experiment (p. 186).

Figure 9.3: Small Scale Blast Experiment



Plan of the small-scale blast experiments as described in [4]. The red circle represents the charge and the green circles represent gauge locations.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

The Autodyn analytical blast simulation was set up to mimic the experimental setup with a Lagrangian volume representing the building with an analytical blast boundary condition applied to the external faces. Gauges were positioned on the volume in the same locations as in the experimental set-up to monitor the applied pressure load using the variable BLAST.LOAD. Additionally, Autodyn Numerical Blast (using the Euler Ideal Gas solver) simulations of the blast were performed, and the pressure load histories (using theory from TM 5-855-1) were computed.

Figure 9.4: Side-On Overpressure (p. 188) and Figure 9.5: Reflected Overpressure (p. 189) compare the side-on overpressure time history and the reflected overpressure time history from the gauge on the front of the building for the Autodyn Numerical Blast model, TM 5-855-1, the Autodyn analytical blast mode,l and the experimental results when the concrete blocks were not present. The peak overpressure and positive phase impulse for these results are summarized in Table 9.1: Peak incident overpressure and positive phase impulses at the gauge on the front of the building for a spherical explosion without the concrete blocks (p. 187) and Table 9.2: Peak reflected overpressure and positive phase impulses at the gauge on the front of the building without the concrete blocks (p. 187). Note that the Autodyn Numerical Blast results agree well with the experimental results and that the analytical blast Autodyn results agree well with the TM 5-855-1 results. The peak incident and reflected overpressures computed by the Autodyn analytical blast model are in reasonable agreement with experiment, although the incident and reflected positive phase impulses are slightly overestimated.

Figure 9.6: Channelled Street Blast Results (p. 189) shows the results from the Autodyn Numerical Blast model, TM 5-855-1, the Autodyn analytical blast model and the experimental results when the concrete blocks were present (a channelled blast). Table 9.3: Peak overpressures and positive phase impulses at both the front and side gauges when the concrete blocks are present (p. 188) summarises the peak overpressure and positive phase impulses for these results. The disagreement between the Autodyn analytical blast results and those from experiment are because the analytical blast formulation does not account for multiple reflections or shadowing by structures.

Table 9.1: Peak incident overpressure and positive phase impulses at the gauge on the front of the building for a spherical explosion without the concrete blocks

	Peak Incident Overpressure (kPa)	Positive Phase Impulse (kPa.ms)
Autodyn Numerical Blast	35.17	9.11
TM 5-855-1	36.49	11.10
Autodyn Analytical Blast	36.60	11.12

Table 9.2: Peak reflected overpressure and positive phase impulses at the gauge on the front of the building without the concrete blocks

	Peak Incident Overpressure (kPa)	Positive Phase Impulse (kPa.ms)
Autodyn Numerical Blast	71.73	80.17
TM 5-855-1	83.65	23.20
Autodyn Analytical Blast	80.17	23.17

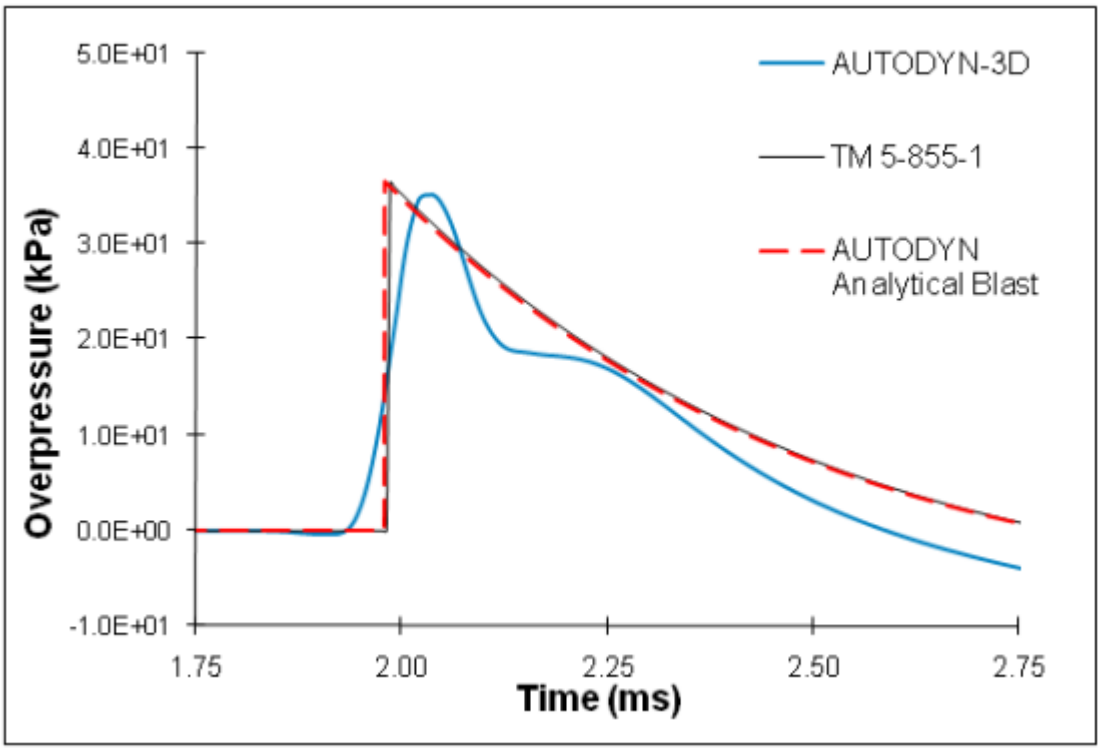
Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

	Peak Incident Overpressure (kPa)	Positive Phase Impulse (kPa.ms)
Experimental	74.72	18.28

Table 9.3: Peak overpressures and positive phase impulses at both the front and side gauges when the concrete blocks are present

	Peak Incident Overpressure (kPa)		Positive Phase Impulse (kPa.ms)	
	Front	Side	Front	Side
Autodyn Numerical Blast	242.9	27.98	63.62	11.68
TM 5-855-1	83.65	-	23.20	-
Autodyn Analytical Blast	80.17	29.33	23.17	9.88
Experimental	295.8	24.74	34.49	8.46

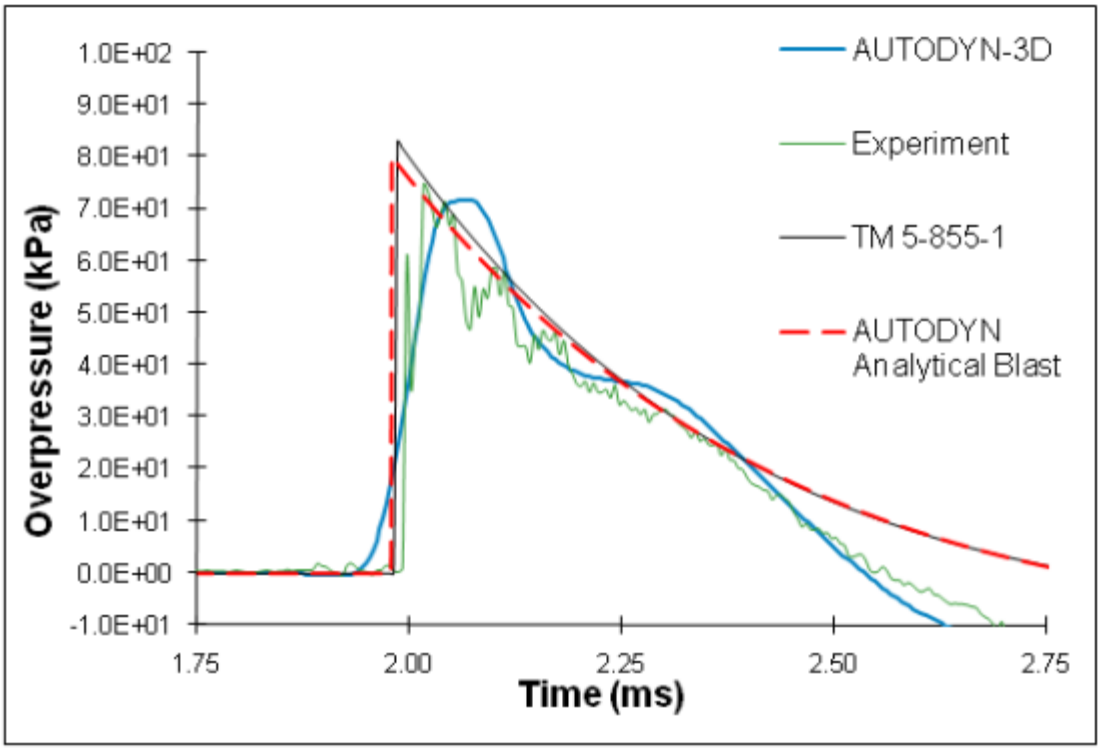
Figure 9.4: Side-On Overpressure



Side-on overpressure experienced 1150 mm away from a 8g TNT charge as predicted by TM 5-855-1, an Autodyn Numerical Blast model analysis, and an Autodyn analytical blast analysis.

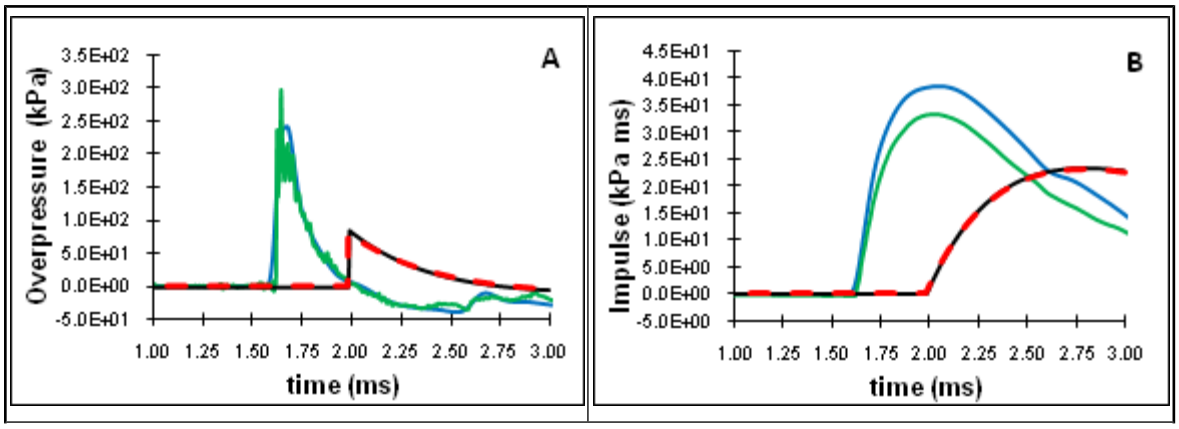
Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 9.5: Reflected Overpressure

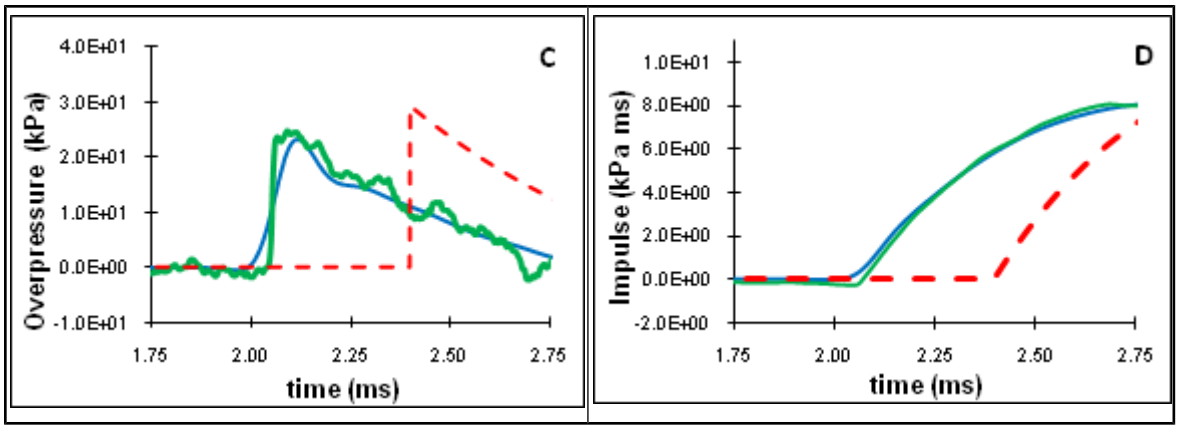


Reflected overpressure at the gauge on the front of the building shown in Figure 9.3: Small Scale Blast Experiment (p. 186). The results from an Autodyn analytical blast analysis are compared to experiment, an Autodyn Numerical Blast model analysis (shown as AUTODYN-3D in the figure), and to the results predicted by TM 5-855-1.

Figure 9.6: Channelled Street Blast Results



Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Channelled street blast results A) overpressure at front gauge B) impulse at front gauge C) overpressure at side gauge D) impulse at side gauge. The results are for the Autodyn Numerical Blast solution (blue), Autodyn Analytical Blast solution (red dashed), TM 5-855-1 (black) and the experimental results (green).

9.3.3. References

The following references are cited in this chapter:

1. Department of the Army (1986). Fundamentals of Protective Design for Conventional Weapons TM 5-855-1, Washington, D.C.

2. Kingery, C. N. and Bulmash, G. (1994) Airblast Parameters from TNT Spherical Air Burst and Hemispherical Surface Burst Technical Report ARBRL-TR-02555, US Army Research and Development Center, Ballistic Research Laboratory.

3. Friedlander, F. G. (1946) The Diffraction of Sound Pulses. I. Diffraction by a Semi-Infinite Plane Proc R. Soc. Lond. A, 186: 322-344.

4. Feng, L. J. (1997) Modelling Blast in an Urban Area Cranfield University Weapons Effects on Structures MSc Project, Royal Military College of Science, Shrivenham, UK.

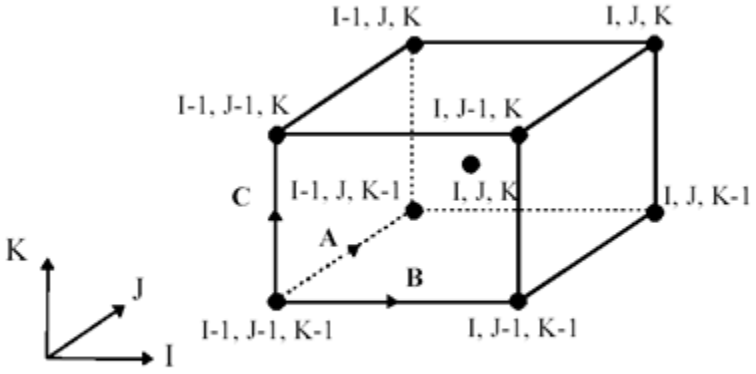
Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Chapter 10: Autodyn Parts

In Autodyn, you build your model in Parts. There are two concepts for node, element and part data storage and processing available.

Structured Parts

For structured parts in Autodyn, all information relating to the mesh and the elements in the mesh is stored and processed using the concept of I, J and K lines (IJK-space). This means that for an element whose quantities are stored at position (I, J, K), the nodes belonging to that element are automatically defined in IJK space as (I-1, J,K), (I, J-1,K), (I-1, J-1,K),.....(see the figure below).



Typical Connectivity for IJK-based Element in AUTODYN

The element connectivity is therefore implicitly defined by the structure within the I, J, K indexing. Information relating to each point in IJK space is also stored in this structure. Furthermore, in this structured approach, the node and element storage and processing is directly linked to the Parts in the model.

Each structured Part is a grid (or group of SPH nodes) that a particular solver operates on.

Autodyn offers many tools for the generation of structured Parts as will be described below.

Unstructured Parts

For unstructured parts an alternative "Unstructured" approach to storing and processing information relating to the mesh is used for the solvers: The basic entities needed in the simulation are the Nodes, Elements, Faces, Joints and so on. In this scheme, these are stored in unique and independent global "Entity" lists, where an entity is a node, element, face, join and so on. Connectivity between entities is determined by cross-referencing to the appropriate global entity list. The data associated with each entity is also stored in a manner most efficient for solving the equations and does not need to follow a set structure/order.

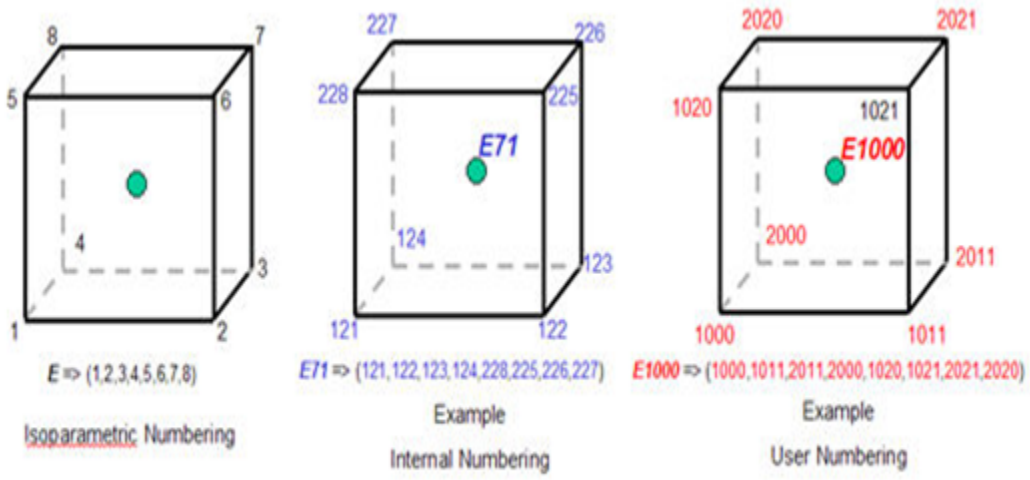
Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

A global list of all elements relating to unstructured Parts is maintained. Each element is given a unique identifying number from one to total number of elements. Each element also has a user element number associated with it for the purposes of model manipulation and user identification.

Each element will have a list of variables associated with it the number and type of variables depending on the type of element (for example a hexahedral element containing an orthotropic material or a triangular shell with a rigid material).

Each element has three numbering systems for its connecting nodes (see figure):

- A local isoparametric number; used to loop over the nodes of an element in the correct order and solved the physical equations. Each element with the same topology has the same numbering system.
- An internal node number; the physical properties of each node are referenced via the global node lists. A connectivity table is used to cross-reference the local node number to an entry in the physical node list.
- A user node number; a number assigned by you to define the element.



Example Element and Node Numbering

For the unstructured solvers in Autodyn, Parts are simply used to allow easy grouping and manipulation of various regions of the model. They are not related to the actual data storage for the elements and nodes, as in IJK-based Parts.

The default definition of a Part currently used for the unstructured solvers is: "A Part is a list of elements of the same classification (volume, shell, beam) that are connected together via the connectivity table of the elements".

Note that within each Part, all elements must be of the same class (volume, shell or beam), but not necessarily the topology. For example a shell Part may contain a mixture of quadrilateral and triangular elements. Nodes are only related to Parts through the element connectivity. Nodes can therefore exist in multiple Parts, for example if the same node is connected to a beam and shell element.

Autodyn doesn't offer any capability of generating unstructured elements and Parts manual. Unstructured elements and Parts can be generated in Ansys Workbench and transferred to Autodyn.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Furthermore unstructured models in MSC.Nastran or LS-Dyna format can also be imported directly into Autodyn.

The following 3D Lagrangian unstructured element types are supported:

- *Volume elements*
  - *8 noded hexahedral elements*
  - *6 noded pentahedral elements (to be used as transition elements)*
  - *4 noded tetrahedral elements*
- *Shell elements*
  - *4 noded quad elements*
  - *3 noded triangular elements (to be used as transition elements)*
- *Beam elements*
  - *2 noded beam elements*

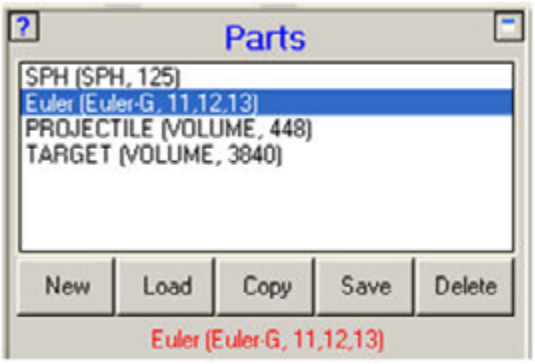
The following 2D Lagrangian unstructured element types are supported:

- *Volume elements*
  - *4 noded quadrilateral volume elements*
  - *3 noded triangular volume elements*

Note:

1. The 2D Lagrangian unstructured element types have Beta status and therefore may not be as stable or robust as expected in a general release version.
2. A 2D or 3D analysis involving unstructured parts with only rigid materials may result in unphysical results, because rigid bodies do not have a minimum time step associated with them and therefore too large displacements may occur from cycle to cycle and contact can be missed. For this type of analysis, you need to manually define a small time step for the analysis.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



This panel lets you create structured **Parts** and modify the structured and unstructured Parts of your model.

Part List

The box at the top of this panel lists the Parts you have currently defined for your model. You can select a Part in this list.

The size of the structured parts is displayed in terms of maximum I, J and K of the part (see Part Euler above). Structured SPH parts are displayed in terms of the total number of SPH nodes, because SPH parts don't use elements.

For unstructured parts the size of part is displayed as the total number of elements (see Parts PROJECTILE and TARGET above).

New

Create a new structured part.

Load

Load a structured Part from a Part Library.

This option is not available for unstructured Parts because for these types of Parts the Ansys Workbench project data storage can be used as a part library.

Only part libraries created in the current version of Autodyn can be imported.

Copy

Copy an existing Part to a new Part. This is valid for both structured as well as unstructured Parts.

Save

Save structured Parts to a Part Library. This option is not available for unstructured Parts because for these types of Parts the Ansys Workbench project data storage can be used as a part library.

Delete

Delete a structured/unstructured Part.

The remaining dialog in this panel depends on the Solver used by the selected Part.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- **All Structured Solvers except SPH**

- **SPH Solver**

- **Unstructured Solvers**

Zoning

Zone a structured Part.

Unstructured Parts cannot be generated; an unstructured Part can only be translated, rotated or scaled.

Fill

Fill a part with materials and initial conditions.

Geometry (Zoning)

Define geometric objects for SPH Parts

Pack (Fill)

Pack geometric objects with SPH nodes.

Boundary

Apply boundary conditions to a Part.

Gauges

Assign gauges to a part.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Motions

Assign motion constraints to a structured ALE Part.

Solver

Select solver options for a Part.

IJK Range

Modify the IJK range for a structured Part

Activation

Set activation / deactivation times for a structured Part.

For unstructured Parts activation / deactivation time cannot be set (they are always activated).

Rename

Rename a Part.

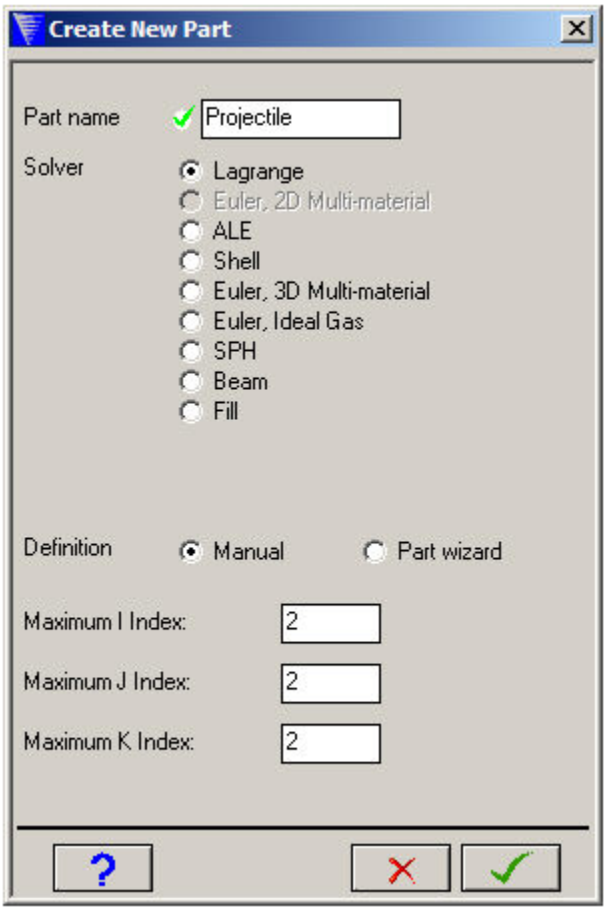
Print

Set print ranges for a Part.

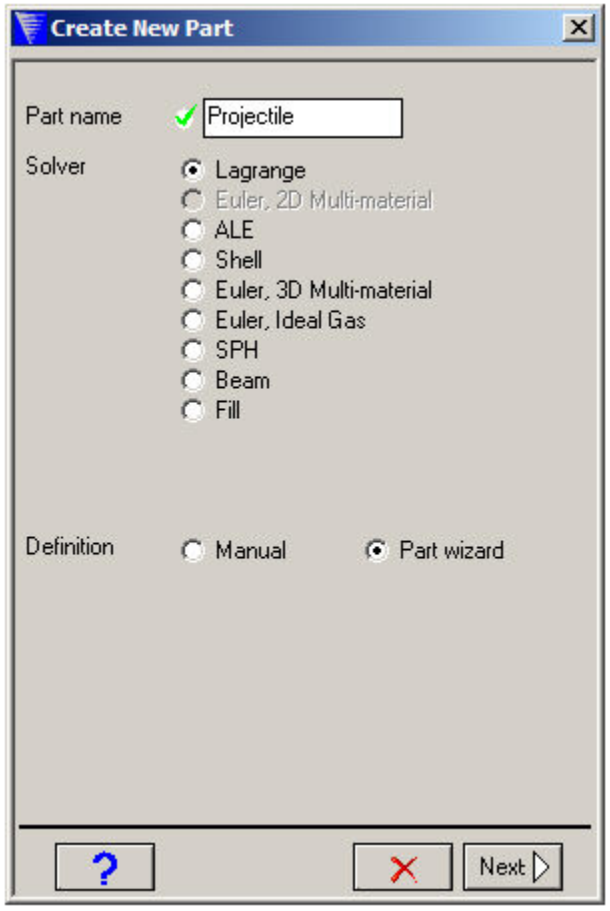
This option is not available for unstructured Parts.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

10.1. Parts - New



Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



This window lets you create a new structured Part.

Part Name

The name for the Part.

Solver

The Solver you want to use for the Part.

Definition

You can select to define your Part manually or using the Part Wizard (the Part Wizard cannot be used for SPH or Beam Solvers, so these options are not displayed for these solvers).

- **Manual**

  If you choose this option, fields appear for you to specify the maximum I, J and K range for your Part (left window above).

  After clicking you return to the Parts panel where you can define your Part manually using the **Zoning** and **Fill** buttons.

- **Part Wizard**

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

If you choose this option, you advance through three more dialog windows that guide you through the complete definition of your part, using one of a number of pre-defined geometries (**Predefs**).

The first dialog window lets you choose the Predef you want to use.

The second dialog window lets you define your zoning.

The third dialog window lets you fill your part.

Click

to advance through the dialog windows, then click in the last window to create your Part.

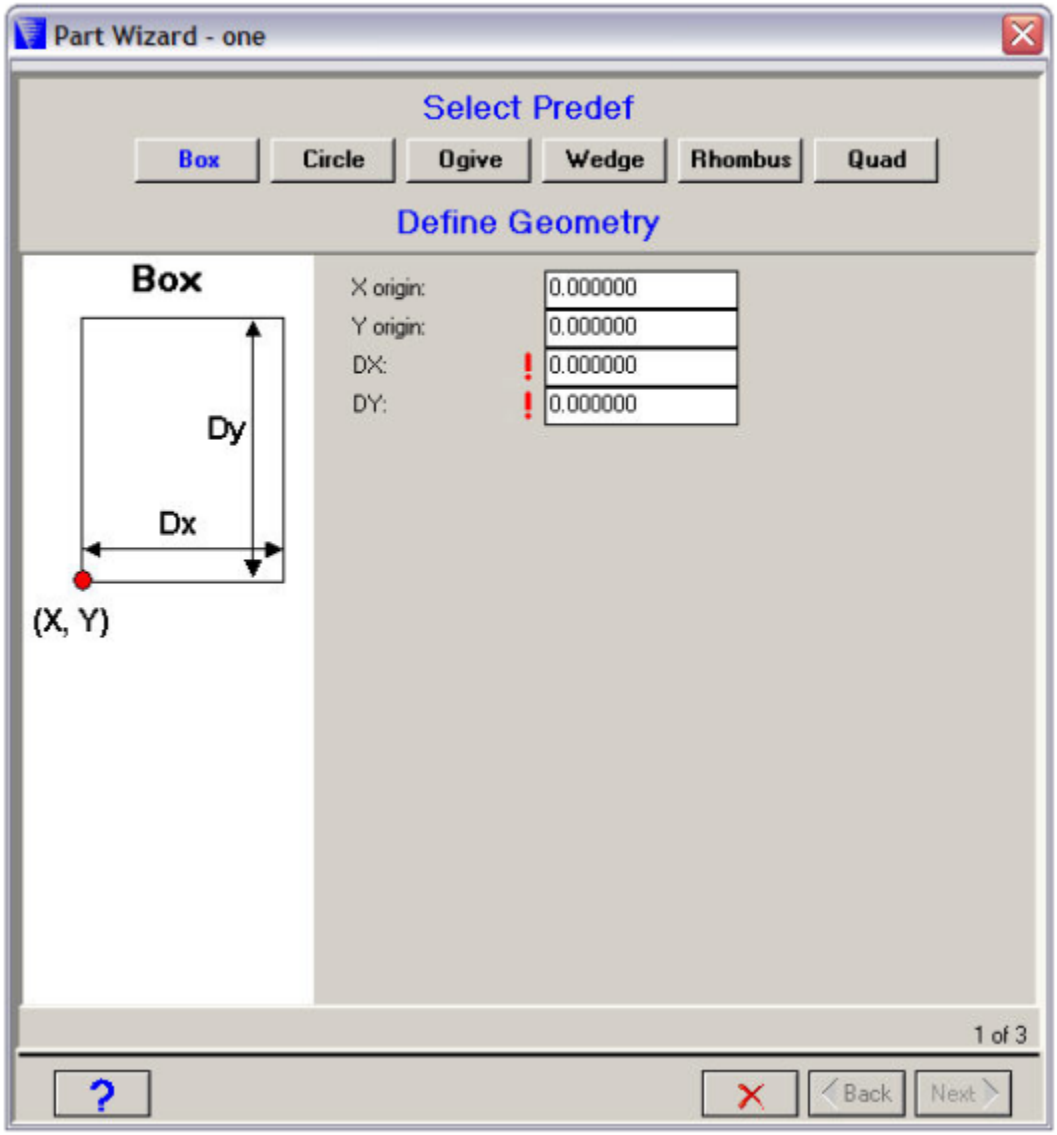
10.1.1. Parts - New - Select Predef (2D)

This window lets you select the Predef you want to use for the 2D Part you are creating.

Click one of the six buttons at the top of this window to select your Predef :

- **Box**

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



This predef is a box with its axes parallel to the X and Y axes.

Origin

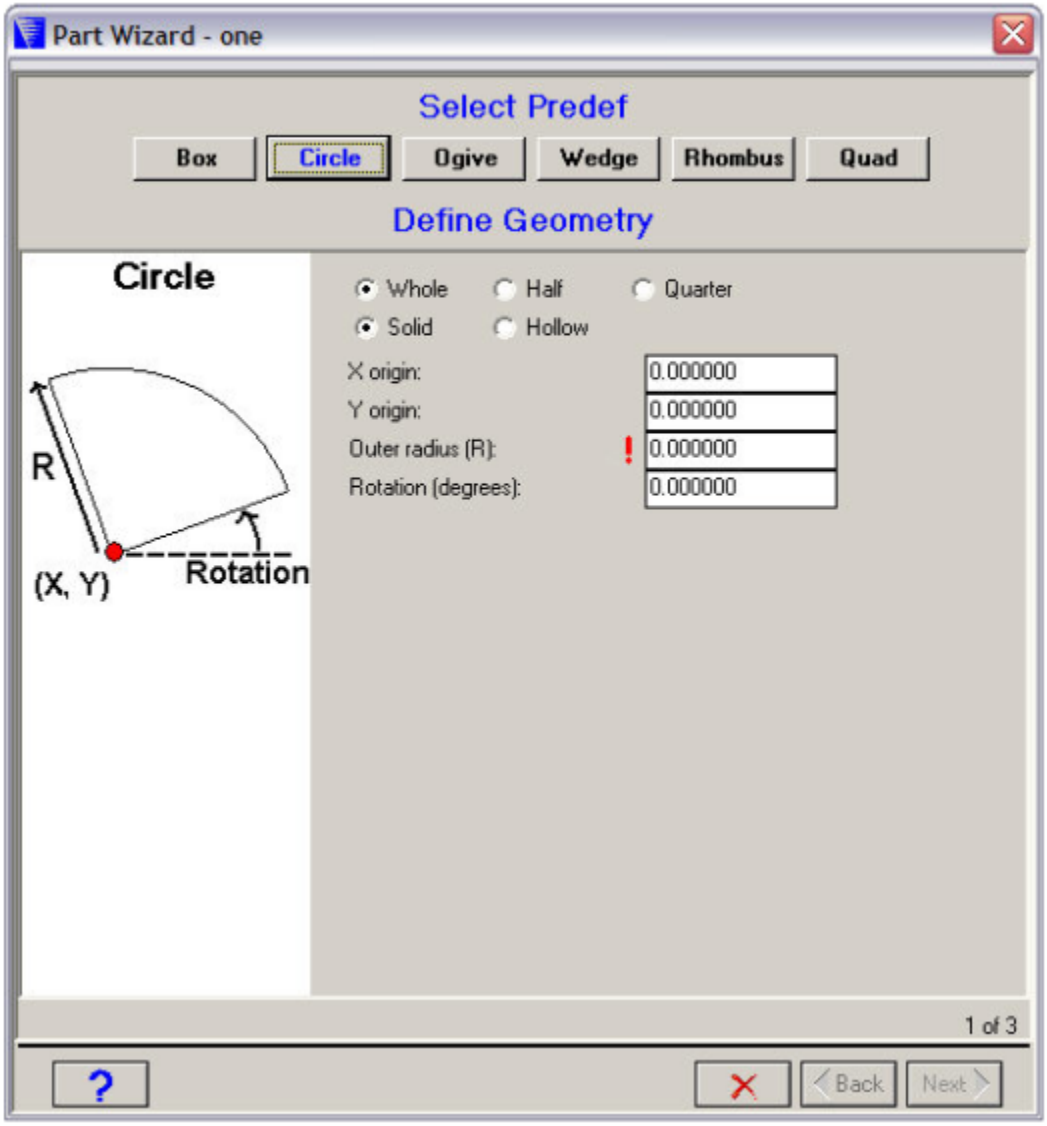
X and Y coordinates of the lower corner of the box.

Box Dimensions

The dimensions of the box (DX, DY).

- **Circle**

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



This predef is a circle.

Section

The section you want to generate (Whole, Half or Quarter). This should match any symmetry planes you have defined for your model.

Solid or Hollow

A solid or a hollow cylinder.

If you choose a hollow cylinder, you will have to specify inner and outer radii for the cylinder.

Origin

The X and Y coordinates for the center of your circle.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

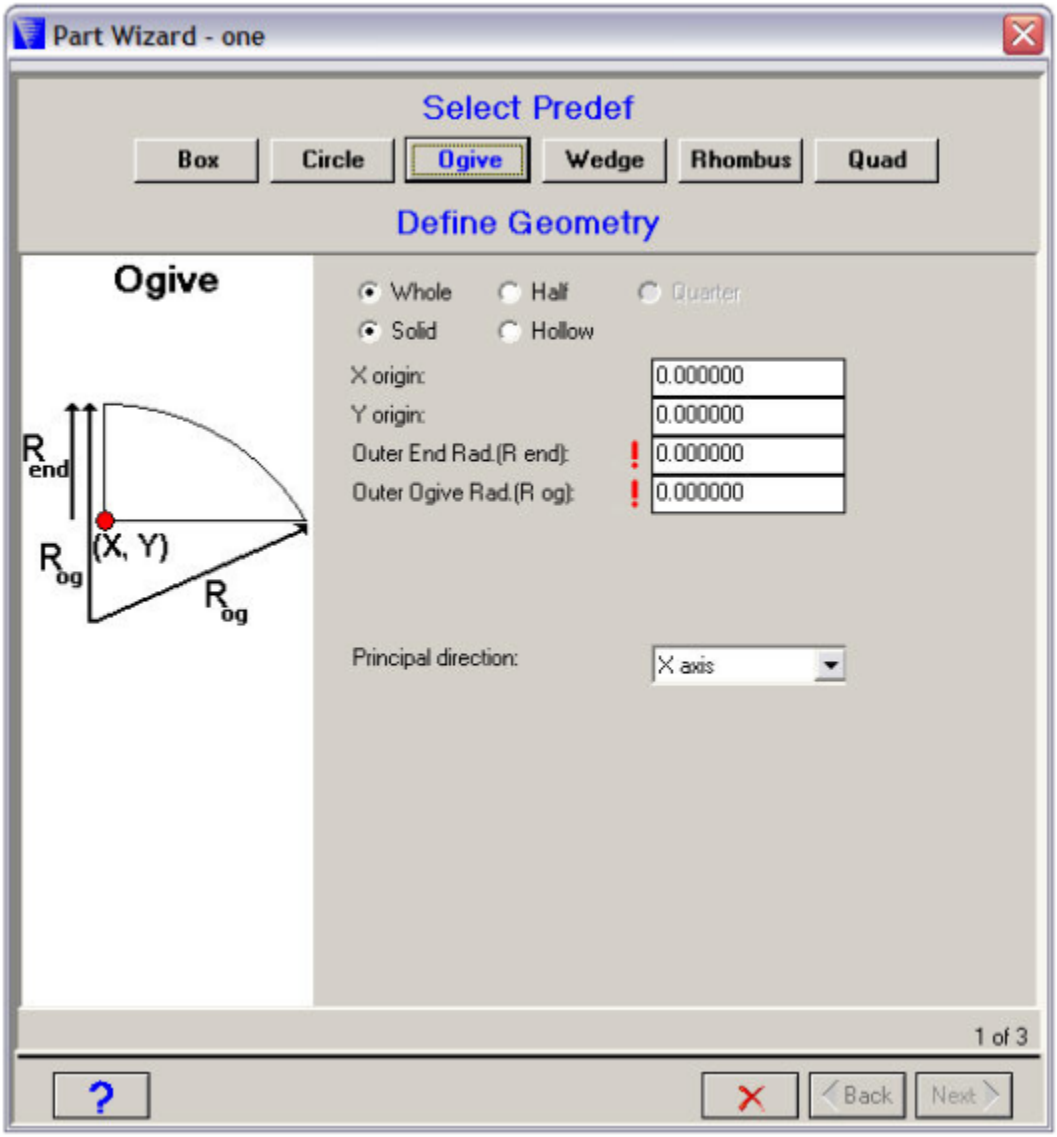
Outer Radius

The outer radius of your circle.

Rotation

The rotation (in degrees) of your circle relative to the X-axis.

- **Ogive**



This predef is an ogive with its axes parallel to the X and Y axes.

Section

The section you want to generate (Whole or Half). This should match any symmetry planes you have defined for your model.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Solid or Hollow

Choose either a solid or hollow ogive.

If you choose a hollow ogive, you will have to specify inner and outer radii for the ogive.

Origin

The X and Y coordinates for the center of your ogive.

End Radius

The end radius of your ogive.

Ogive Radius

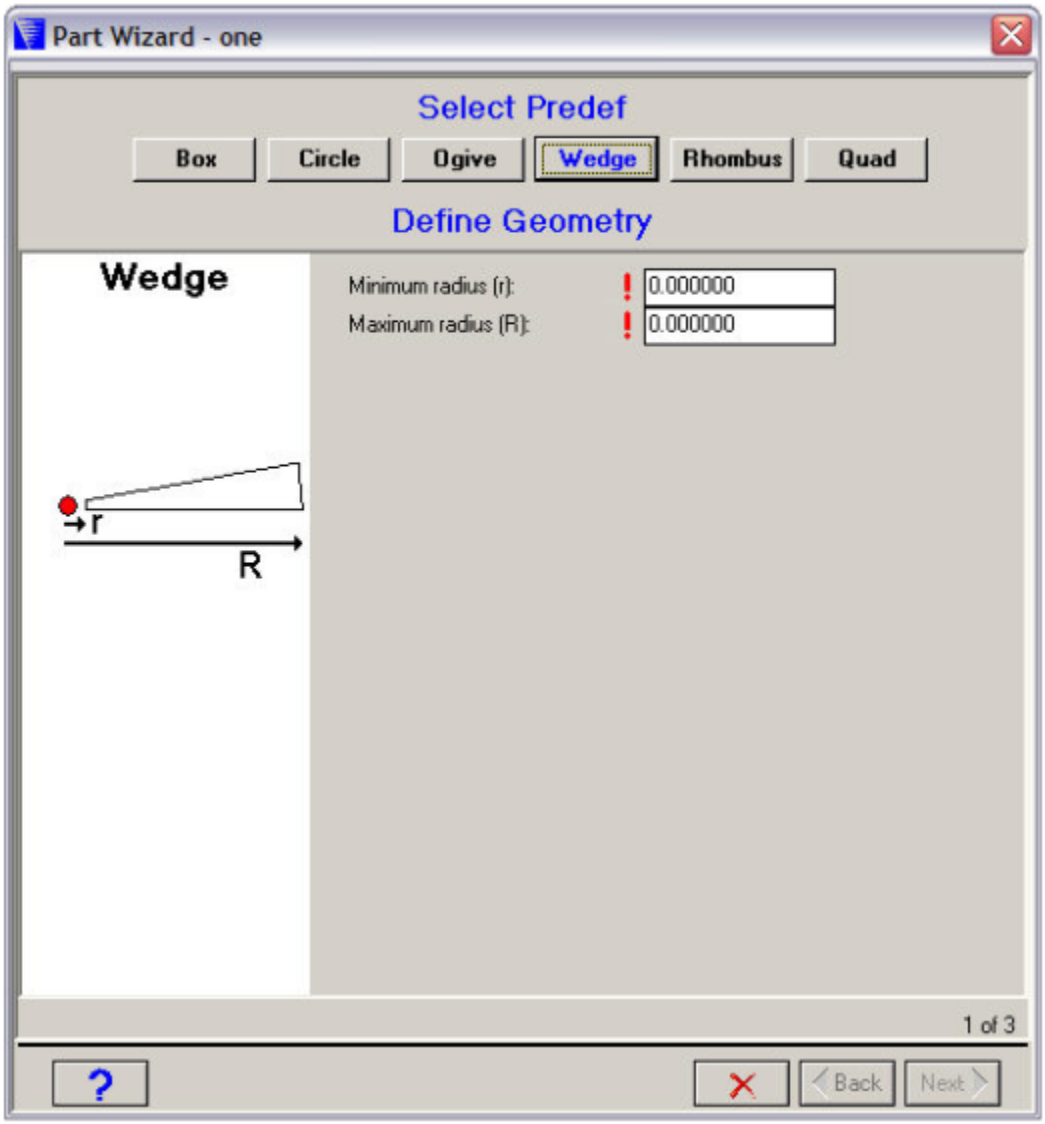
The ogive radius.

Principal Direction

The direction of the principal axis for your ogive.

- **Wedge**

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



This predef is a wedge with its lower edge lying on the X-axis. The wedge angle is automatically set by Autodyn.

This predef is useful for running 1D cylindrical or spherical symmetric problems in 2D.

Minimum radius

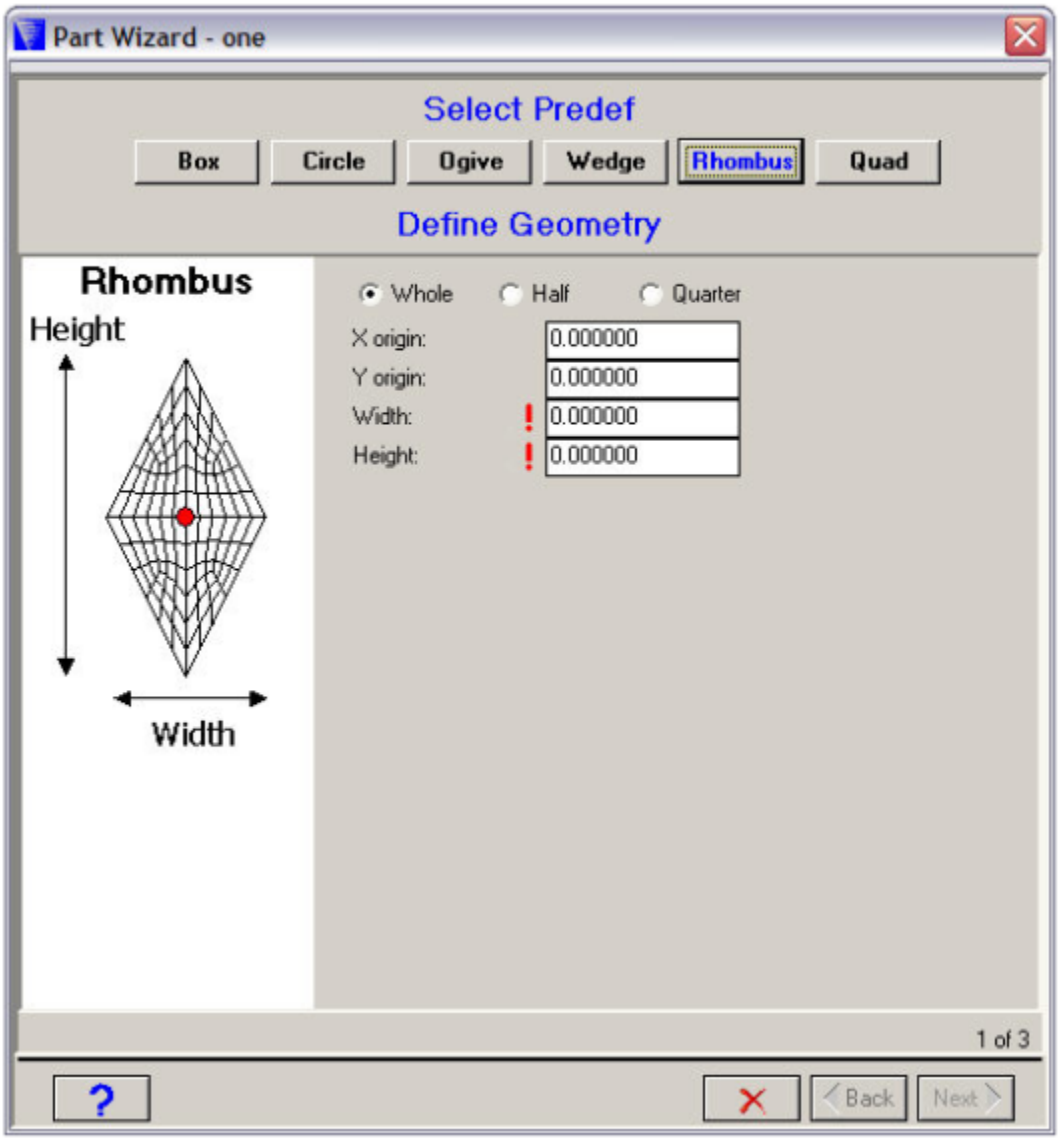
The minimum radius of the wedge (this cannot be zero).

Maximum radius

The maximum radius of the wedge.

- **Rhombus**

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



This predef is a rhombus with its diagonals parallel to the X and Y axes.

Section

The section you want to generate (Whole, Half or Quarter). This should match any symmetry planes you have defined for your model.

Origin

The X and Y coordinates for the center of your rhombus.

Width

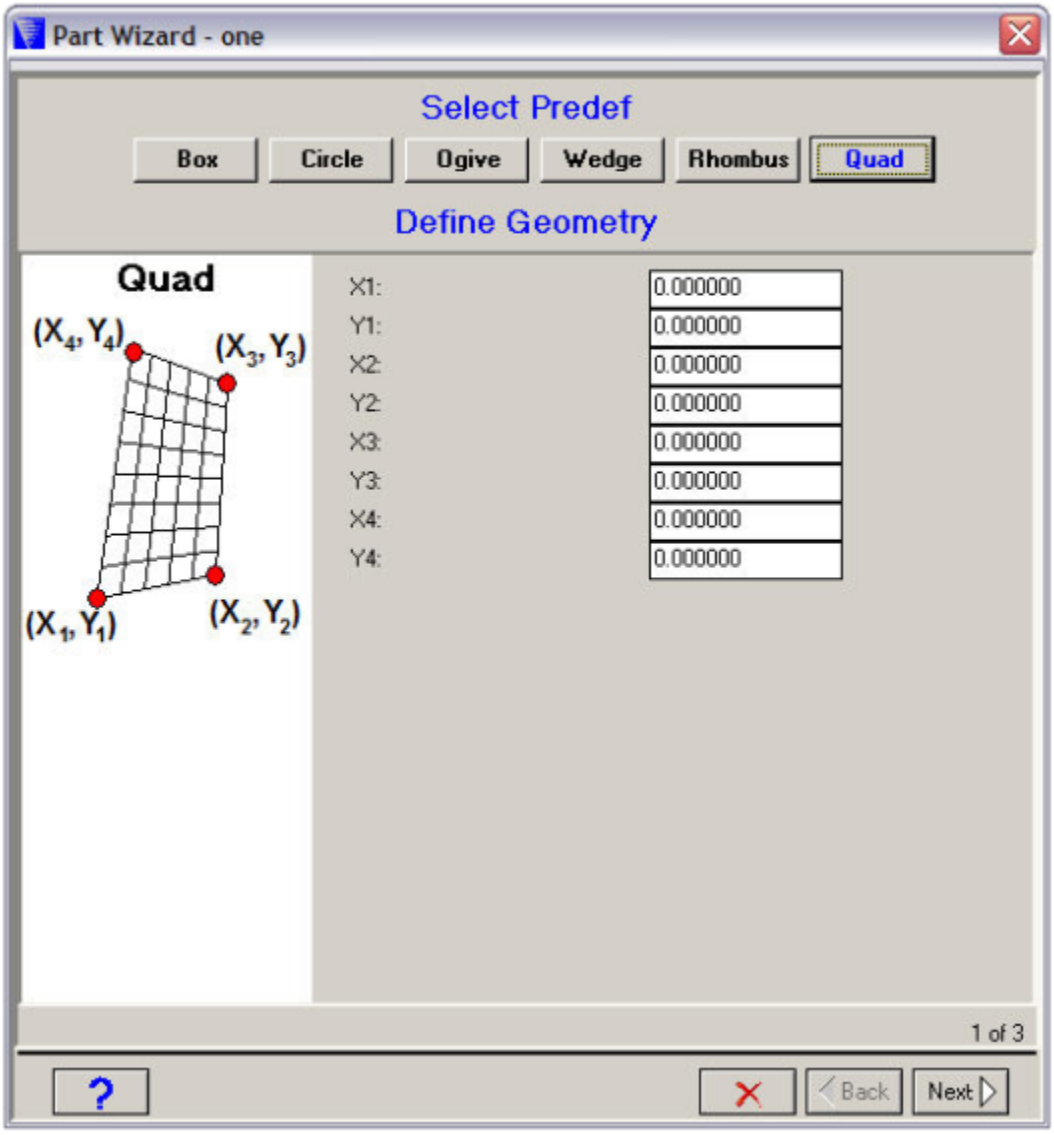
The width of the rhombus.

Height

The height of the rhombus.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- **Quad**



Coordinates X1, Y1

Coordinates of the first point of the quad.

Coordinates X2, Y2

Coordinates of the second point of the quad.

Coordinates X3, Y3

Coordinates of the third point of the quad.

Coordinates X4, Y4

Coordinates of the fourth point of the quad.

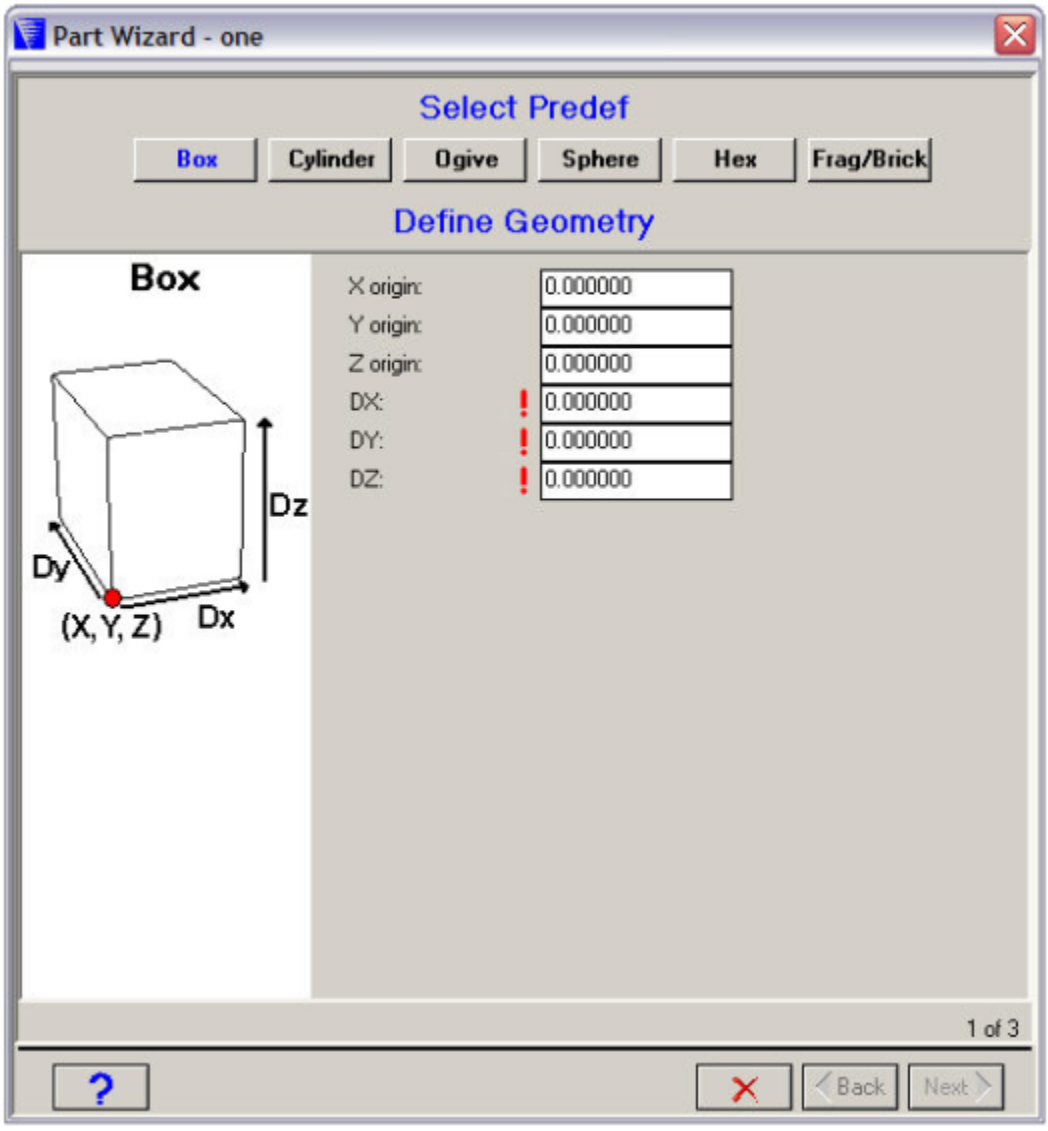
Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

10.1.2. Parts - New - Select Predef (3D)

This window lets you select the Predef you want to use for the 3D Part you are creating.

Click one of the six buttons at the top of this window to select your Predef :

- **Box**



This predef is a box with its axes parallel to the X and Y axes.

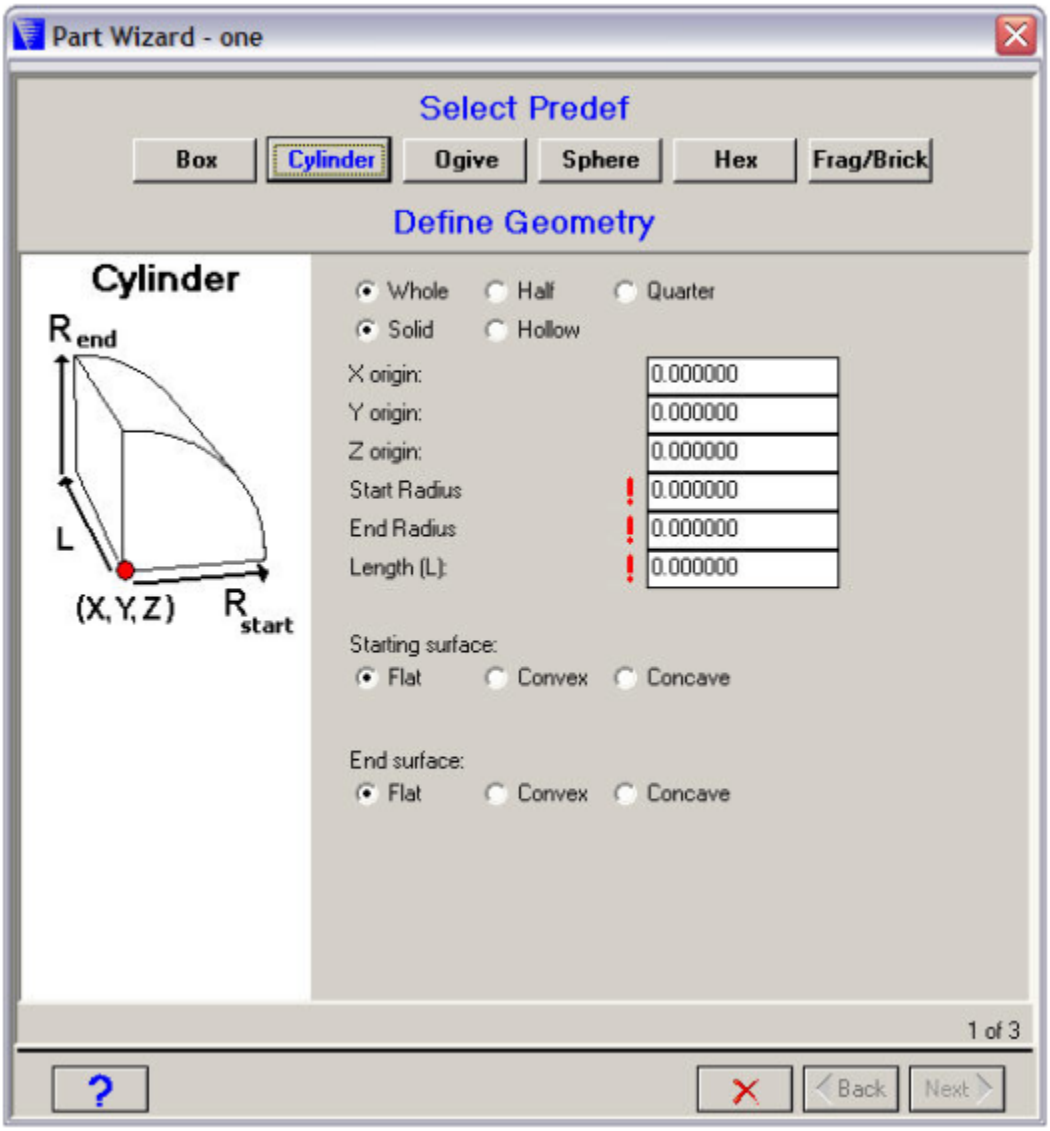
Origin

The X, Y, and Z coordinates of the lower corner of the box.

Box Dimensions

The dimensions of the box (DX, DY, DZ).

- **Cylinder**



This predef is a cylinder with its axis parallel to the Z axis.

Section

The section you want to generate (Whole, Half or Quarter). This should match any symmetry planes you have defined for your model.

Solid or Hollow

Choose either a solid or hollow cylinder.

If you choose a hollow cylinder, you will have to specify inner and outer radii for the cylinder.

Origin

The X, Y, and Z coordinates for the center of the starting surface of your cylinder.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Start Radius / End Radius

You can specify different radii for the starting and end faces of your cylinder. For a regular cylinder (constant radius), set the starting radius equal to the end radius.

Length

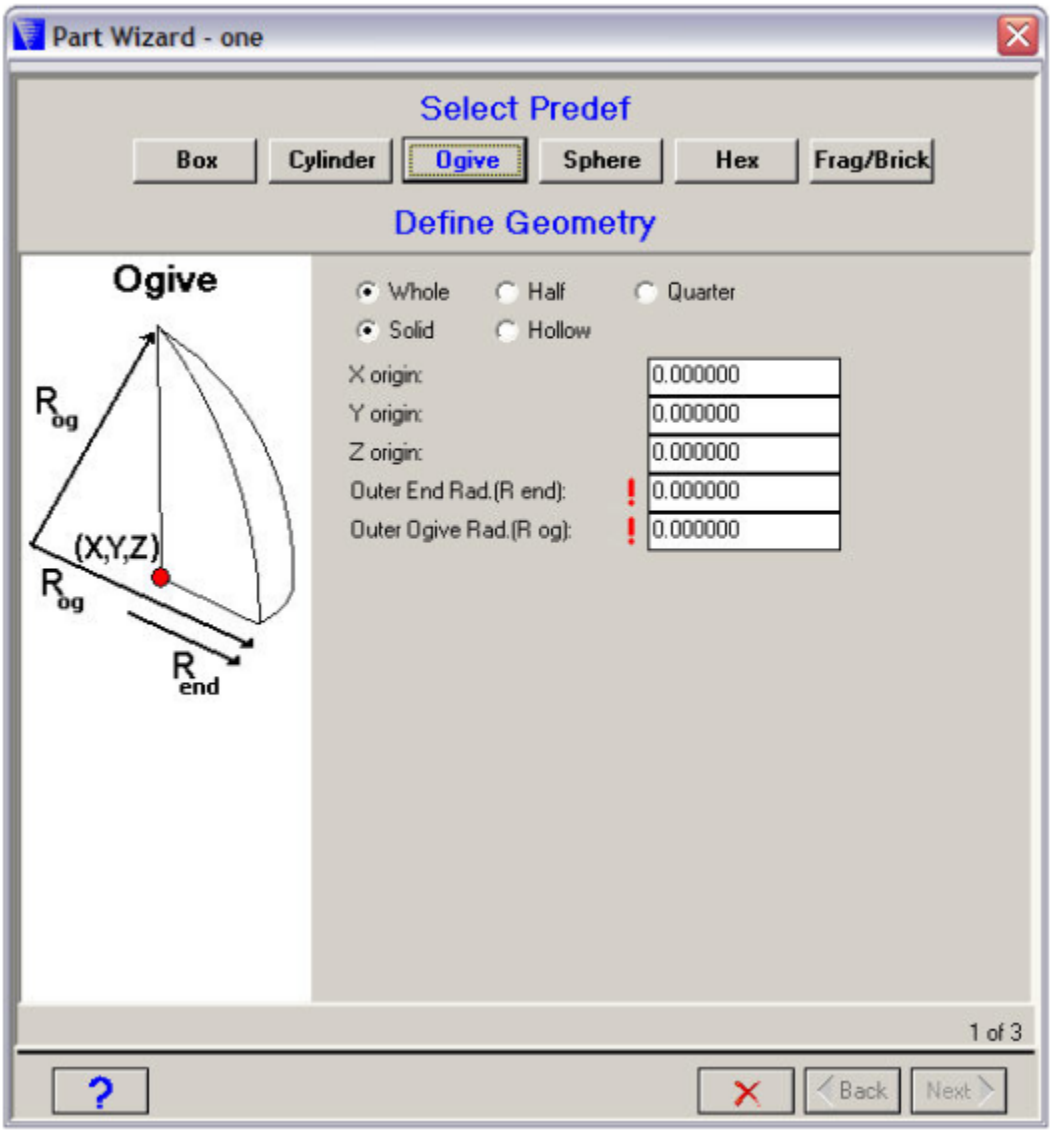
The length of your cylinder.

Start Surface / End surface

The starting and end surfaces of your cylinder can be flat, concave or convex.

If you select a concave or convex surface, you will have to specify its radius of curvature.

- **Ogive**



This predef is an ogive with its axis parallel to the Z axis.

Section

The section you want to generate (Whole, Half or Quarter). This should match any symmetry planes you have defined for your model.

Solid or Hollow

Choose either a solid or hollow ogive.

If you choose a hollow ogive, you will have to specify inner and outer radii for the ogive.

Origin

The X, Y, and Z coordinates for the center of your ogive.

End Radius

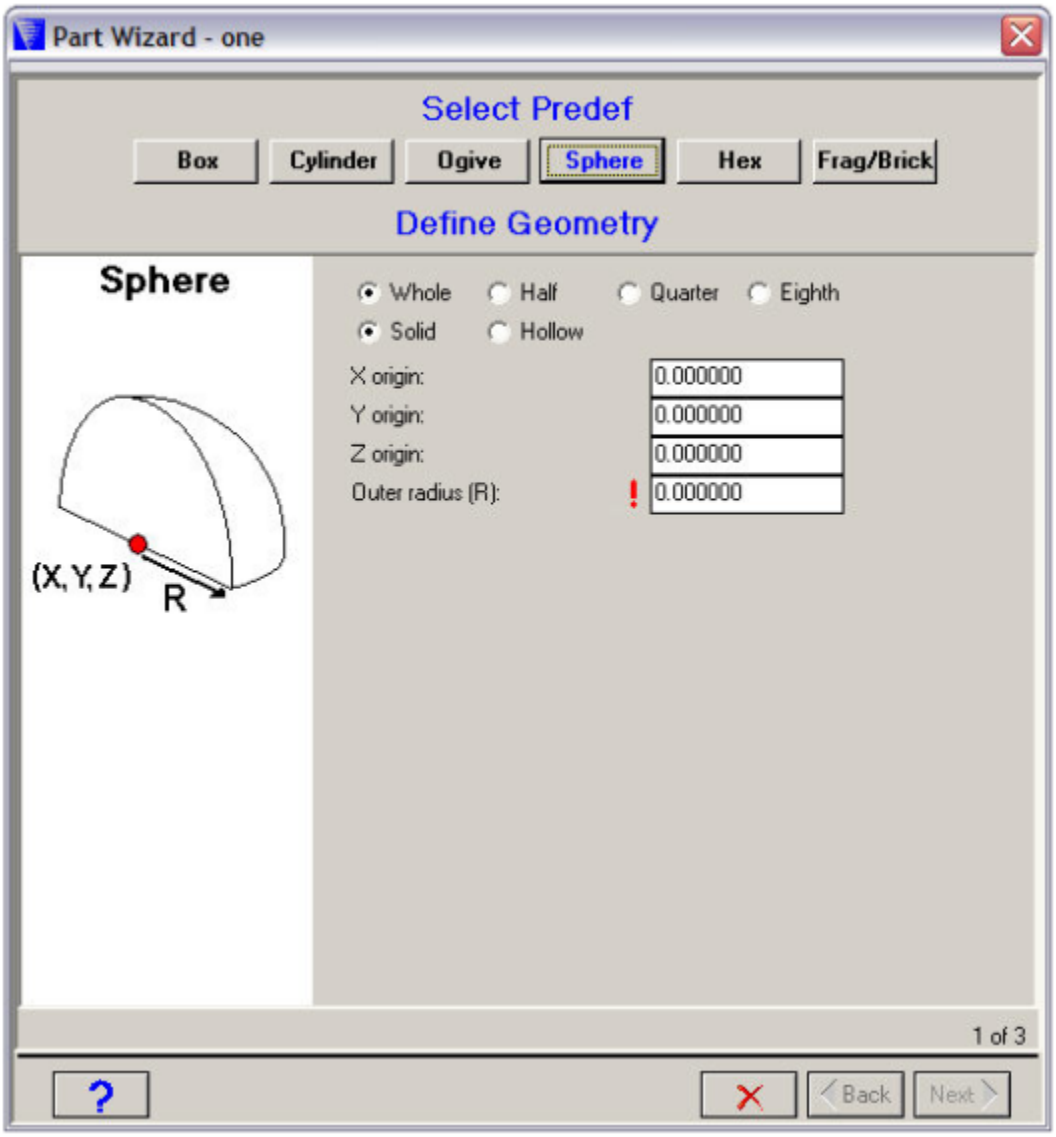
The end radius of your ogive.

Ogive Radius

The ogive radius.

- **Sphere**

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



This predef is a sphere.

Section

The section you want to generate (Whole, Half, Quarter or Eighth). This should match any symmetry planes you have defined for your model.

Solid or Hollow

Choose either a solid or hollow sphere.

If you choose a hollow sphere, you will have to specify inner and outer radii for the sphere.

Origin

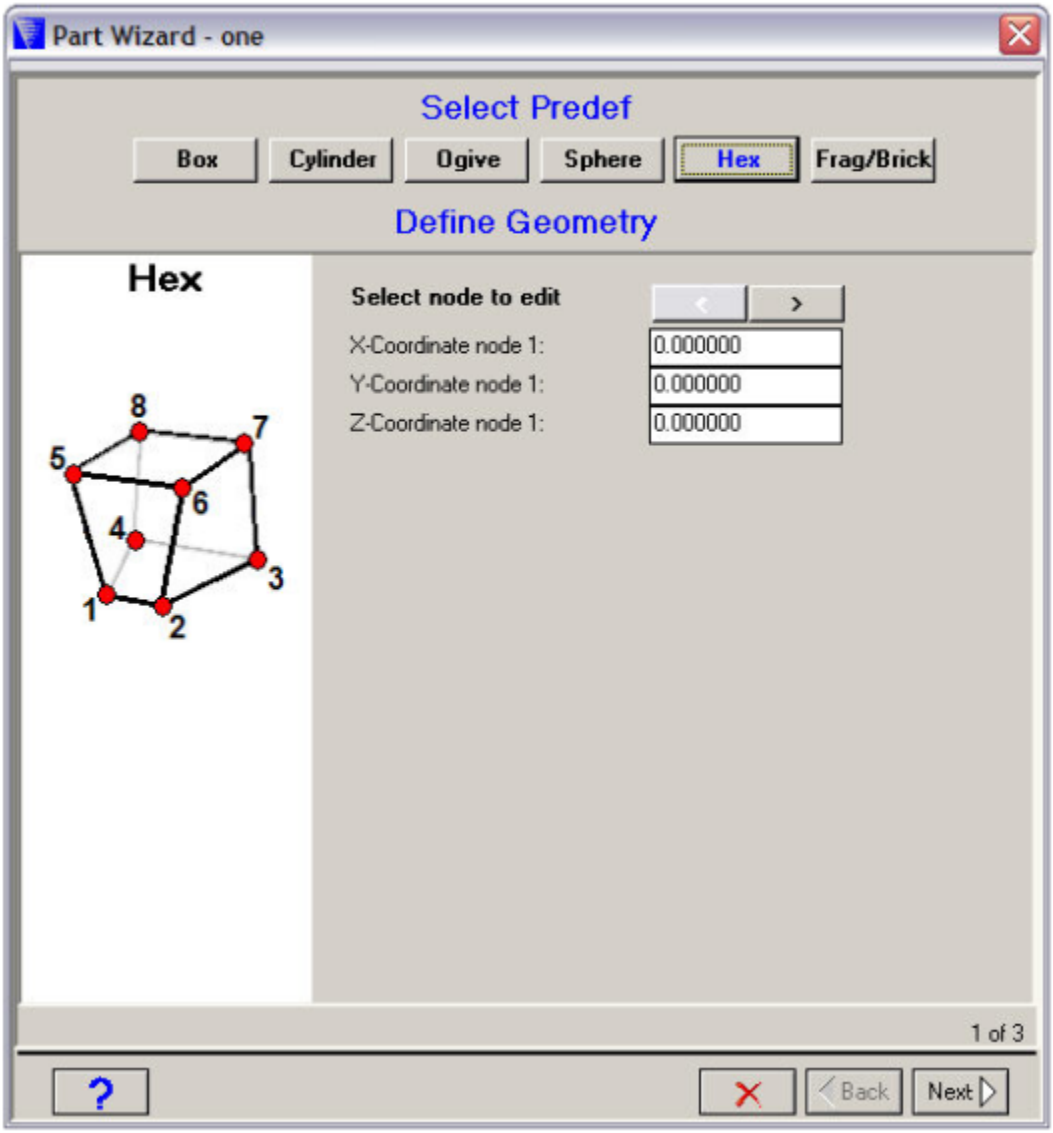
The X, Y, and Z coordinates for the center of the sphere.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Radius

The radius of the sphere.

- **Hex**



Select Node to Edit

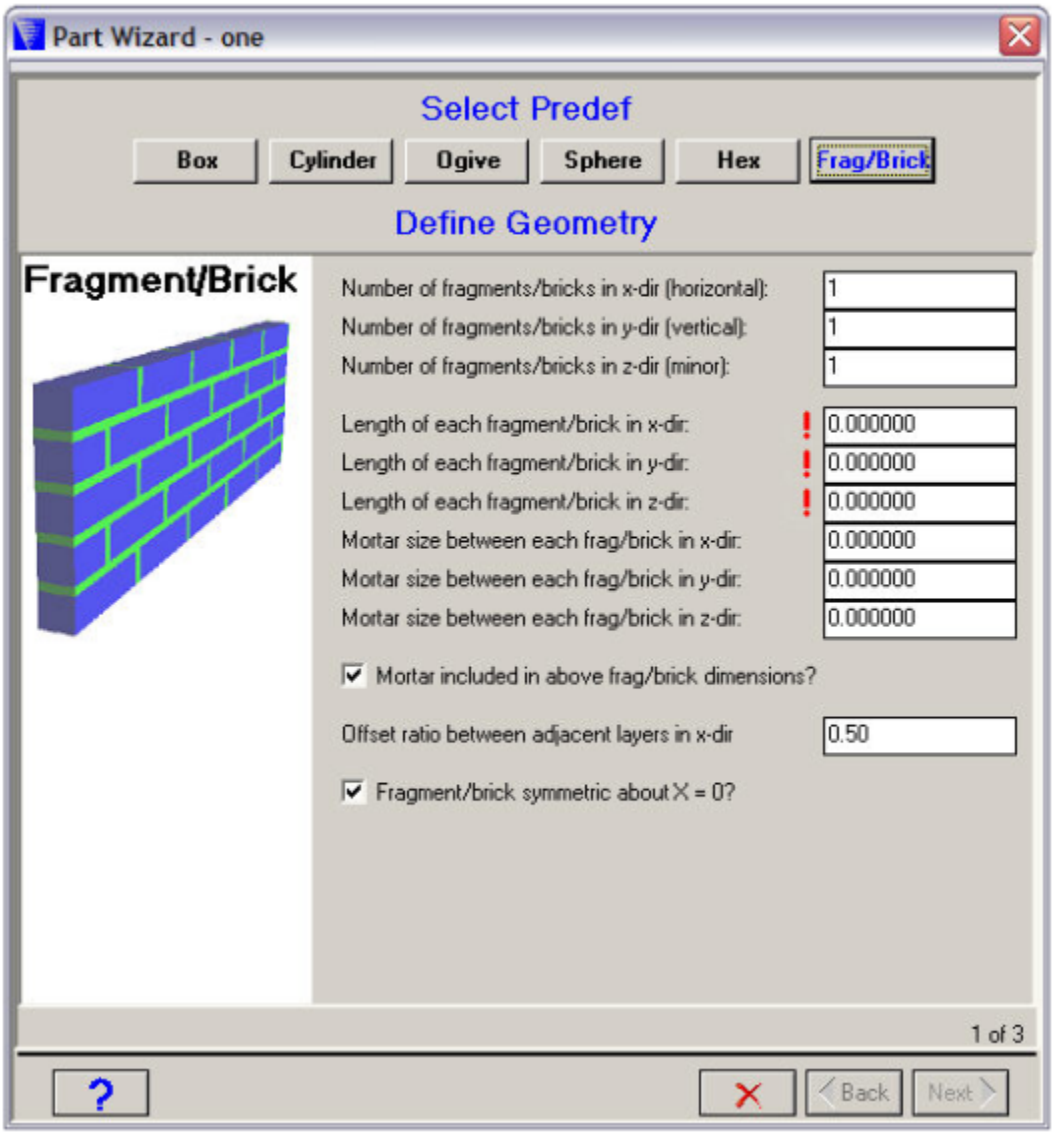
Use the arrow buttons to select each of the eight nodes in turn.

Coordinates

The X, Y, and Z coordinates of the node.

- **Frag/Brick**

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



The fragment/brick wall is built with its axes parallel to the X, Y, and Z axes.

The X direction runs the length of the wall.

The Y direction runs the height of the wall.

The Z direction runs through the thickness of the wall.

Number of fragments/bricks

The number of fragments/bricks in the X, Y, and Z directions.

Length of each fragment/brick

The length of a single brick/fragment in the X, Y, and Z directions.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Mortar size between each fragment/brick

The mortar size (or gap) between fragments/bricks.

Mortar included in dimension?

Check this box if you want the mortar size to be included in the length of each brick/fragment.

Offset ratio

The offset ratio for the fragments/bricks.

This is how the bricks are staggered along the length of the wall. For example, if the offset is 0.5 (the default), the bricks will be staggered halfway across each other as shown in the above figure.

Fragment/brick symmetric about X=0?

Check this box if you want the wall to be symmetric about X=0.

Enter the required dimensions for your predef and click

to advance to the next dialog window (**Define Zoning**).

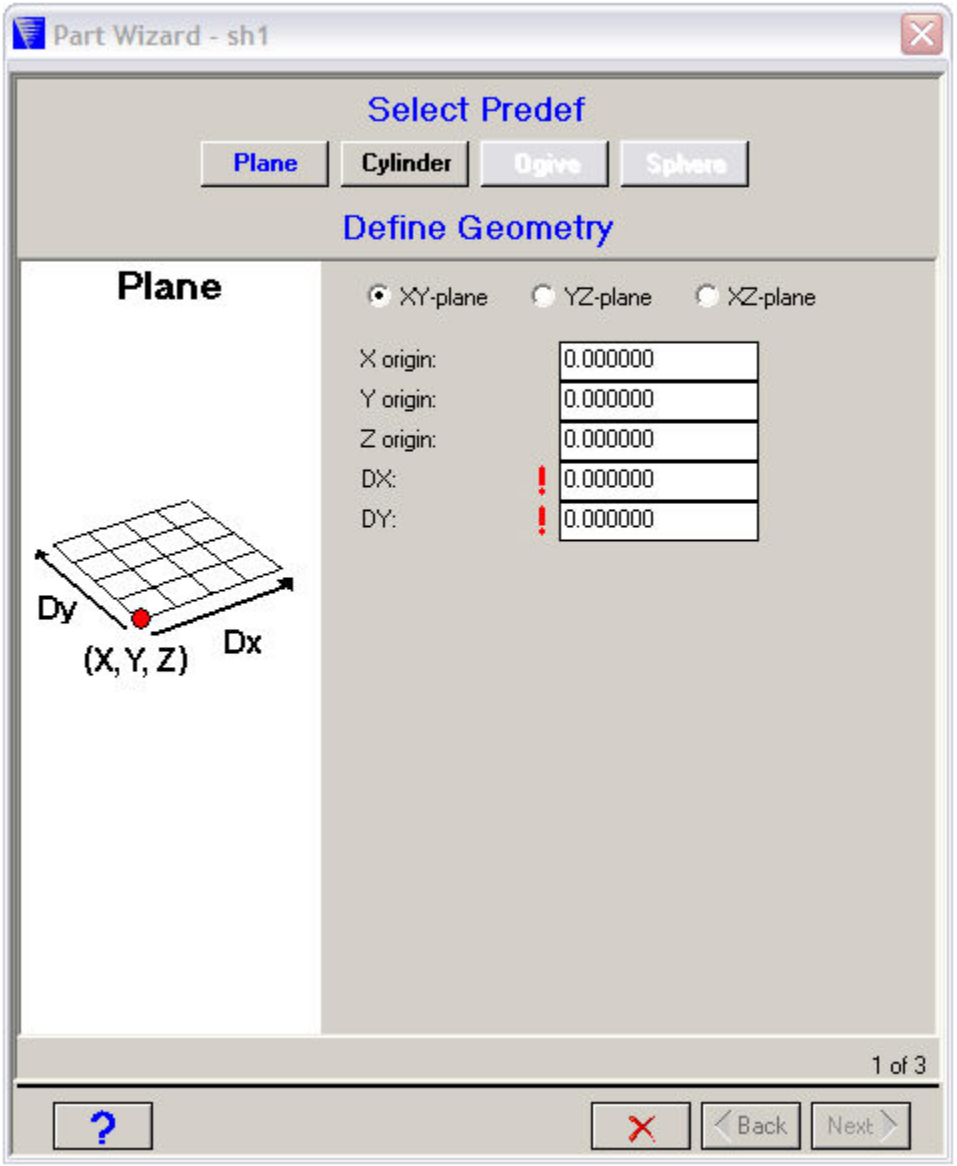
10.1.3. Parts - New - Select Predef (3D Shell)

This window lets you select the Predef you want to use for the Shell Part you are creating.

Click one of the two buttons at the top of this window to select your Predef:

- **Plane**

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



This predef is a plane perpendicular to the X, Y, or Z axis.

Plane Orientation

The orientation of the plane (e.g. XY-plane is a plane perpendicular to the Z axis).

Origin

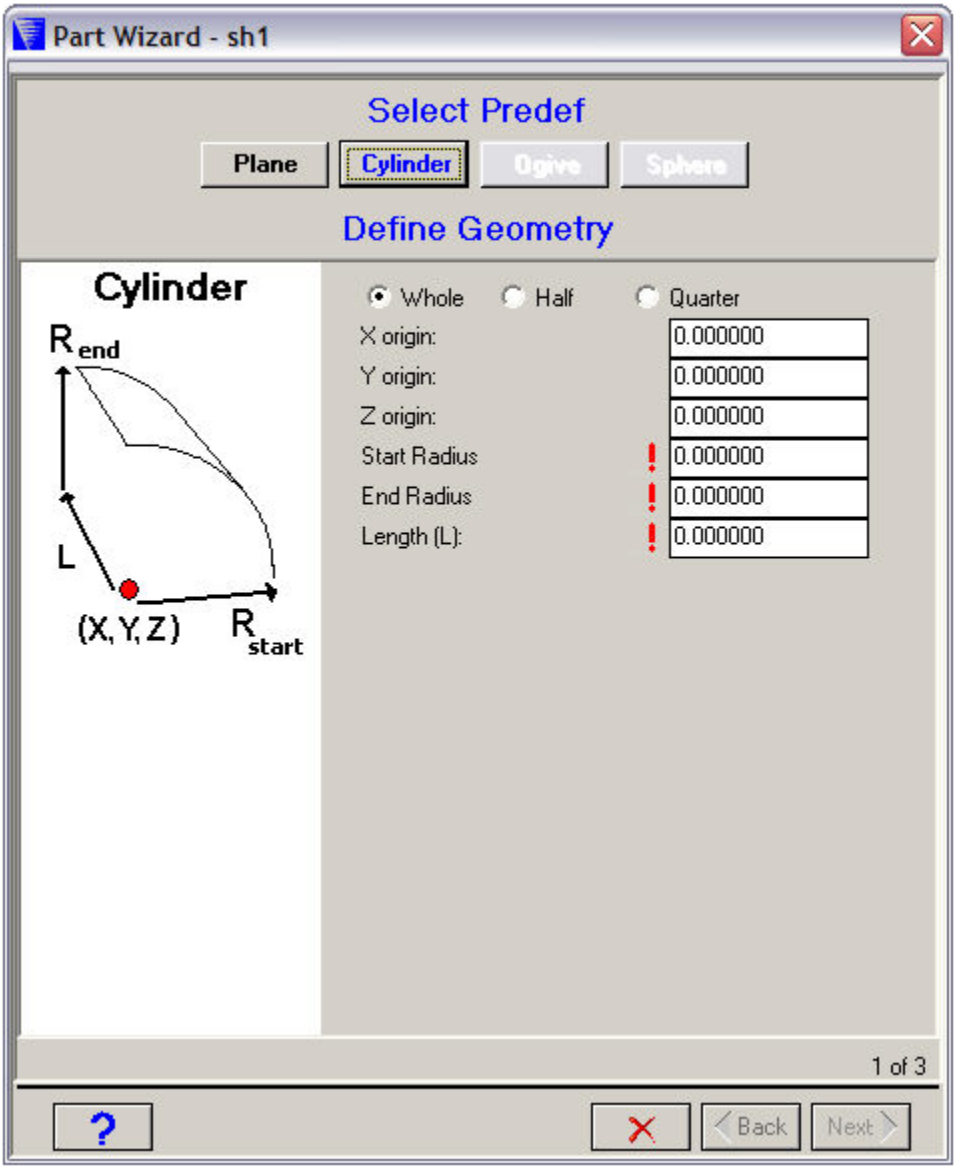
The X, Y, and Z coordinates of the lower corner of the plane.

Box Dimensions

The dimensions of the box (DX, DY, DZ).

- **Cylinder**

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



This predef is a cylinder with its axis parallel to the Z axis.

Section

The section you want to generate (Whole, Half or Quarter). This should match any symmetry planes you have defined for your model.

Origin

The X, Y, and Z coordinates for the center of the starting surface of your cylinder.

Start Radius / End Radius

You can specify different radii for the starting and end faces of your cylinder. For a regular cylinder (constant radius), set the starting radius equal to the end radius.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

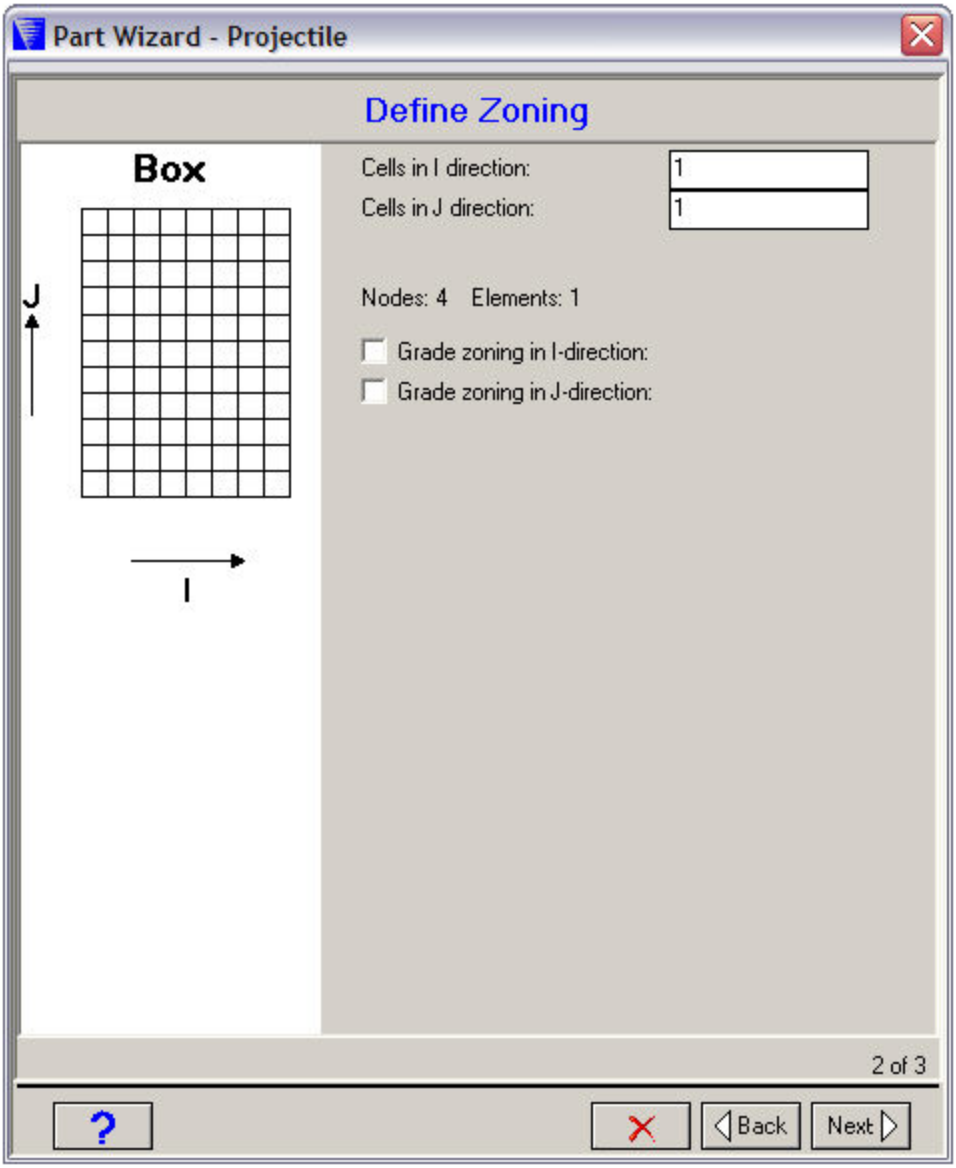
Length

The length of your cylinder.

10.1.4. Parts - New - Define Zoning (2D)

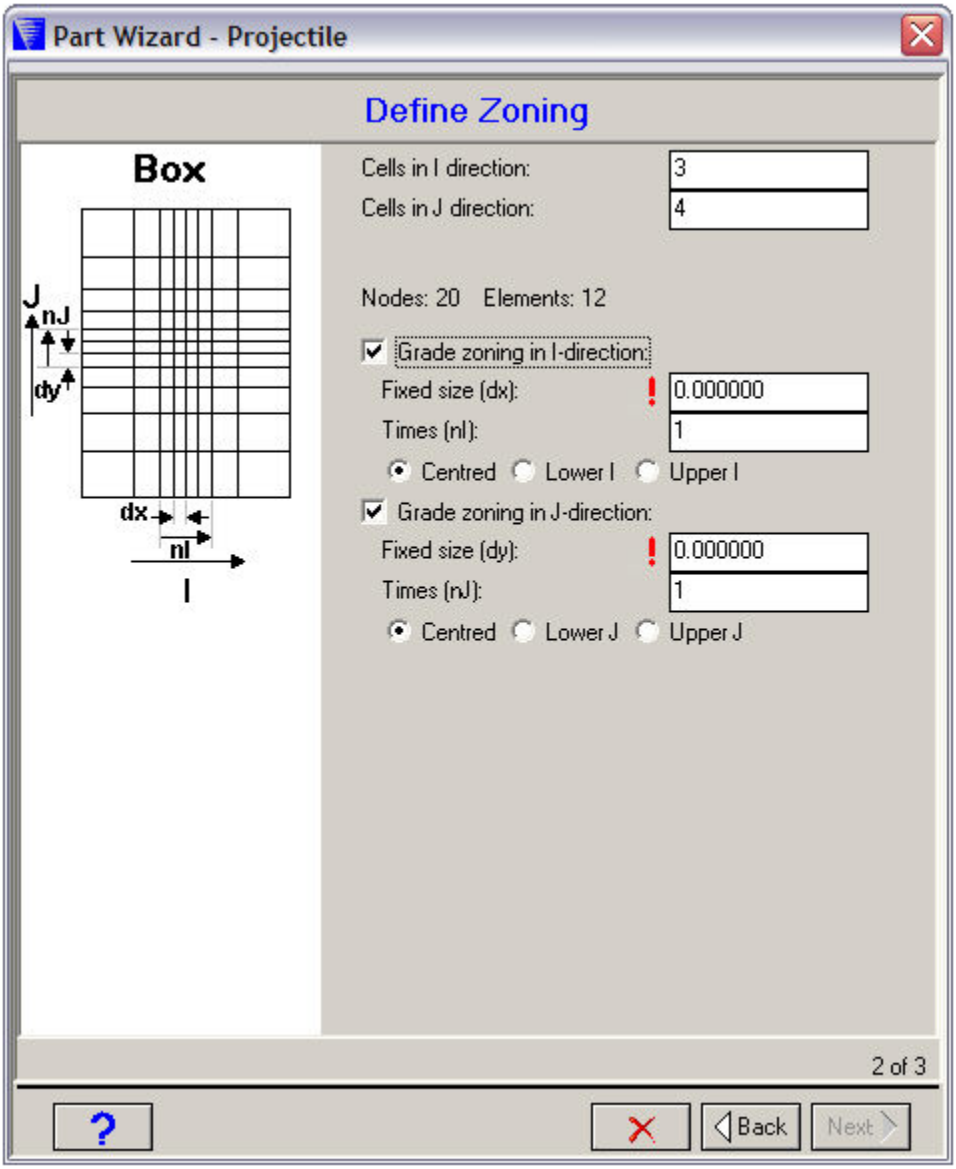
This window lets you Define the zoning for your 2D Part.

- **Box**



No Grading

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Grading in X and Y

Cells

The number of cells (elements) you want to use in the I and J directions. The corresponding number of nodes and elements required for your part will be displayed below.

Grade zoning

Check the boxes if you want to grade your zoning in any direction.

If you choose this option you can specify a range of cells that have a fixed size. The remaining cells will be smoothly graded to fit the overall dimension of the box you specified in the previous window.

– **Fixed size**

The dimension you want to use for the fixed size cells.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

– **Times**

The number of fixed size cells you want to use.

– **Position**

Specify where you want to place the fixed size cells relative to the full I/J range.

Centered - at the center of the full I/J range

Lower - at the lower end of the I/J range

Upper - at the upper end of the I/J range

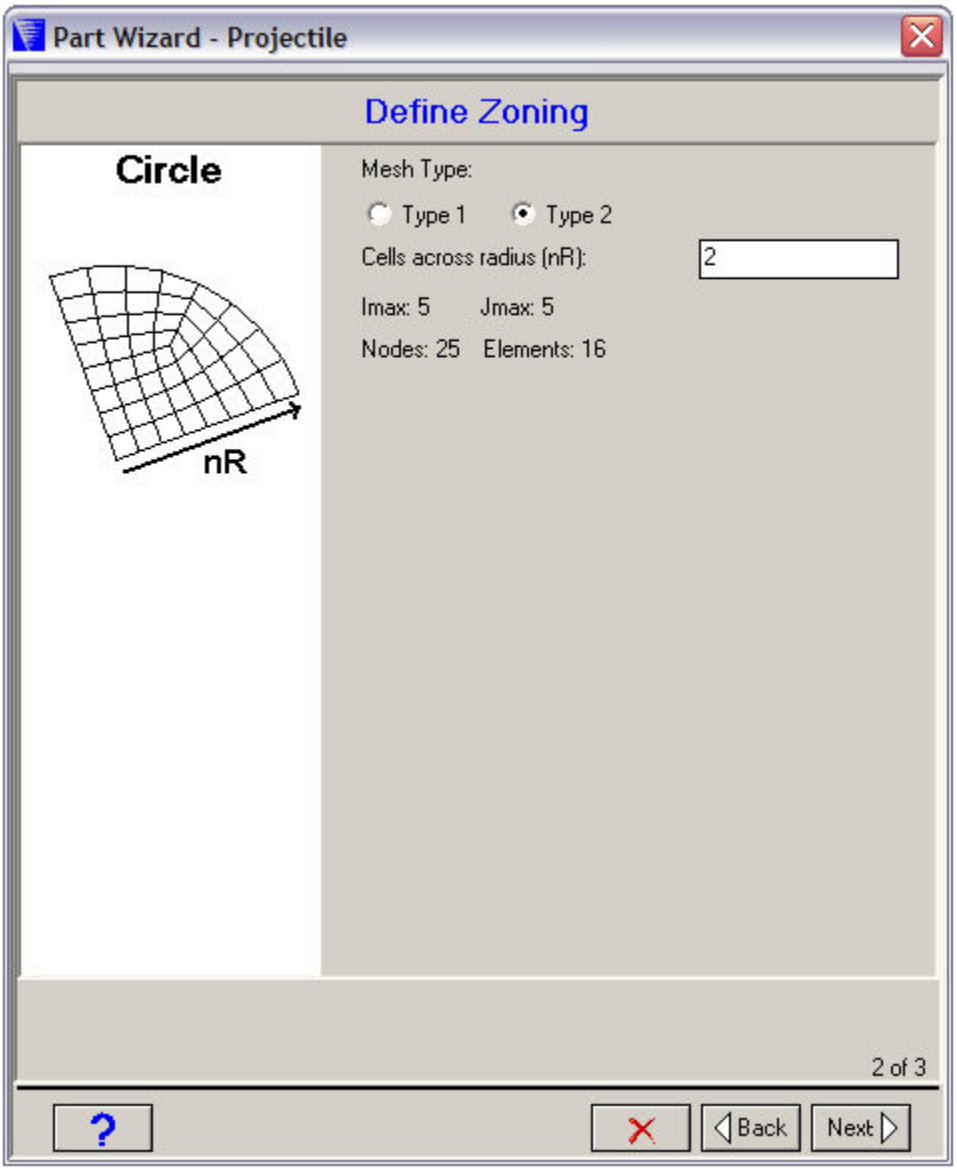
e.g. ten fixed size cells centered :

e.g. twelve fixed size cells at the lower end :

e.g. eight fixed size cells at the upper end :

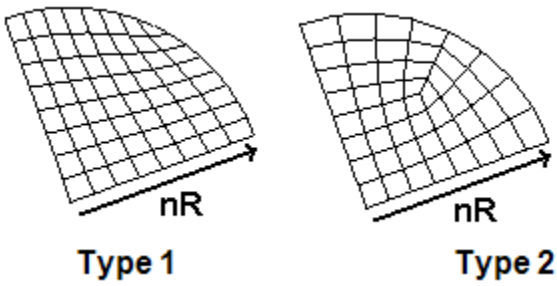
- **Circle**

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Mesh Type

Select a Type 1 or Type 2 mesh for your circle.



The type 1 mesh is more efficient in using elements, but some elements are not very rectangular (which can reduce the accuracy of your solution).

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

All the elements of a type 2 mesh are reasonably rectangular, but about a quarter of the elements generated for this mesh are unused.

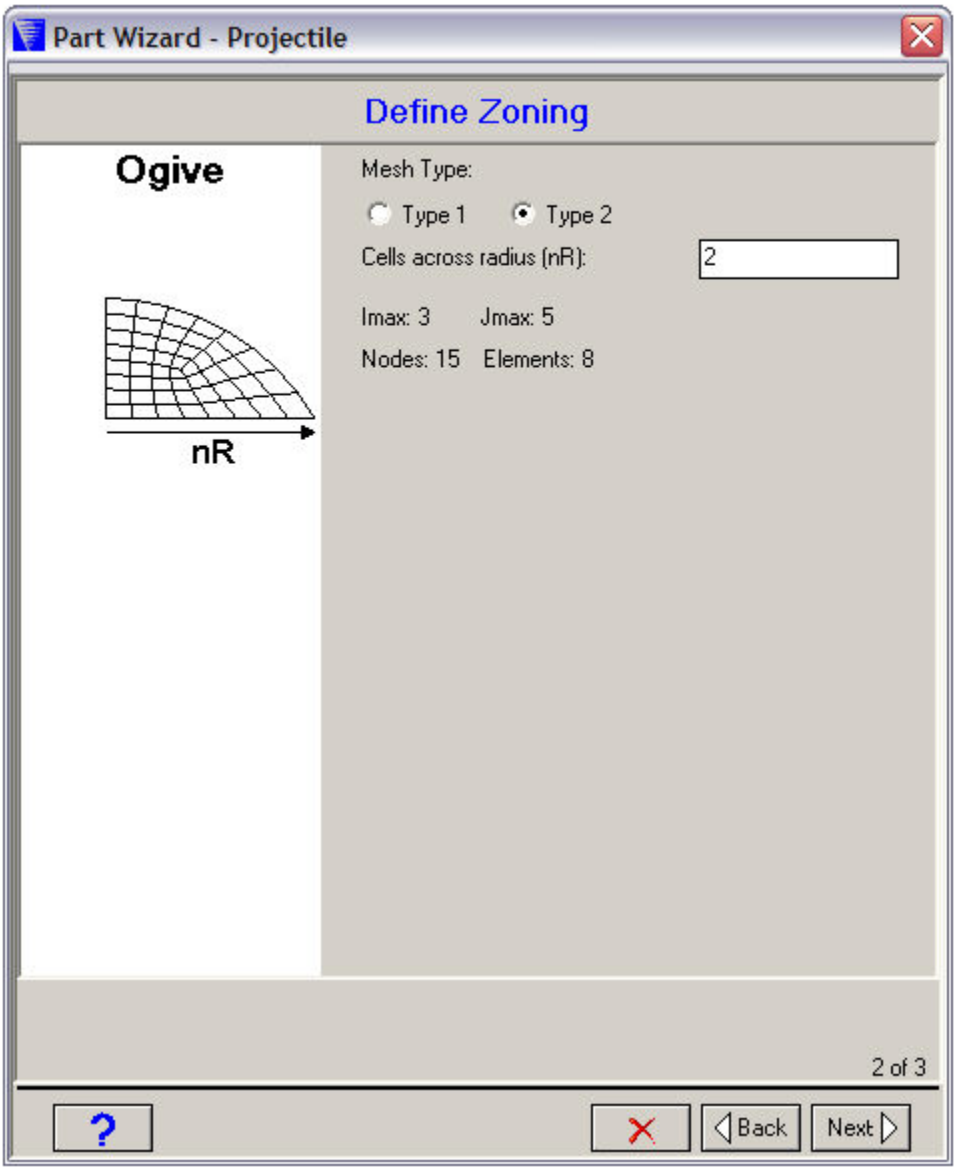
Cells across radius

The number of cells (elements) you want to use across the radius of your circle.

Number of elements

Displayed below the input fields is the index space that will be used for your Part and the number of nodes and elements required.

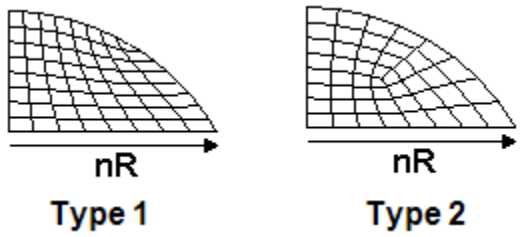
- **Ogive**



Mesh Type

Select a Type 1 or Type 2 mesh for your ogive.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



The type 1 mesh is more efficient in using elements, but some elements are not very rectangular (which can reduce the accuracy of your solution).

All the elements of a type 2 mesh are reasonably rectangular, but some of the elements generated for this mesh are unused.

Cells across radius

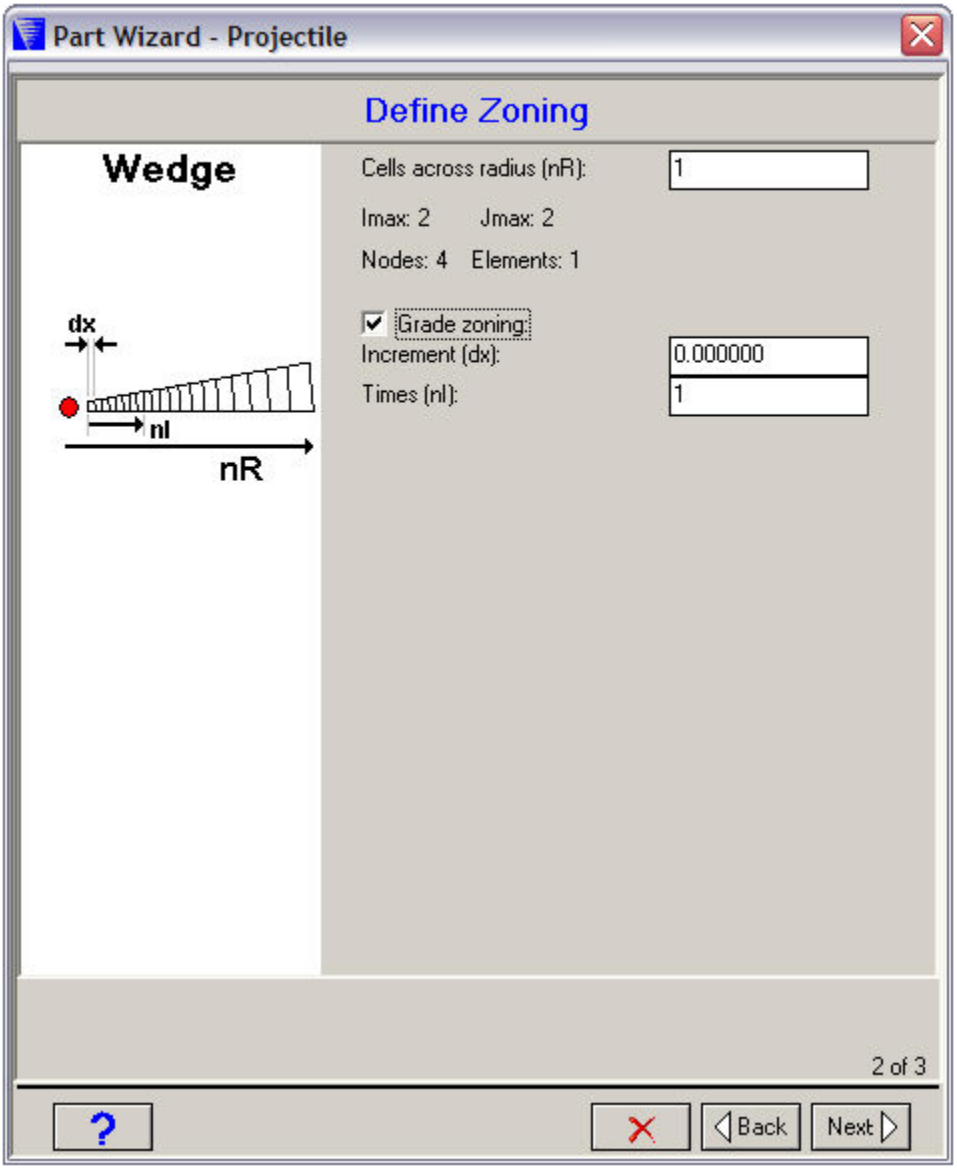
The number of cells (elements) you want to use across the radius of your ogive.

Number of elements

Displayed below the input fields is the index space that will be used for your Part and the number of nodes and elements required.

- **Wedge**

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Cells across radius

The number of cells (elements) you want to use across the radius of your wedge.

Number of elements

Displayed below the first input field is the index space that will be used for your Part and the number of nodes and elements required.

Grade zoning

Check this box if you want to grade your zoning.

If you choose this option you can specify a range of cells which have a fixed size. The remaining cells will be smoothly graded to fit the overall dimension of the box you specified in the previous window.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

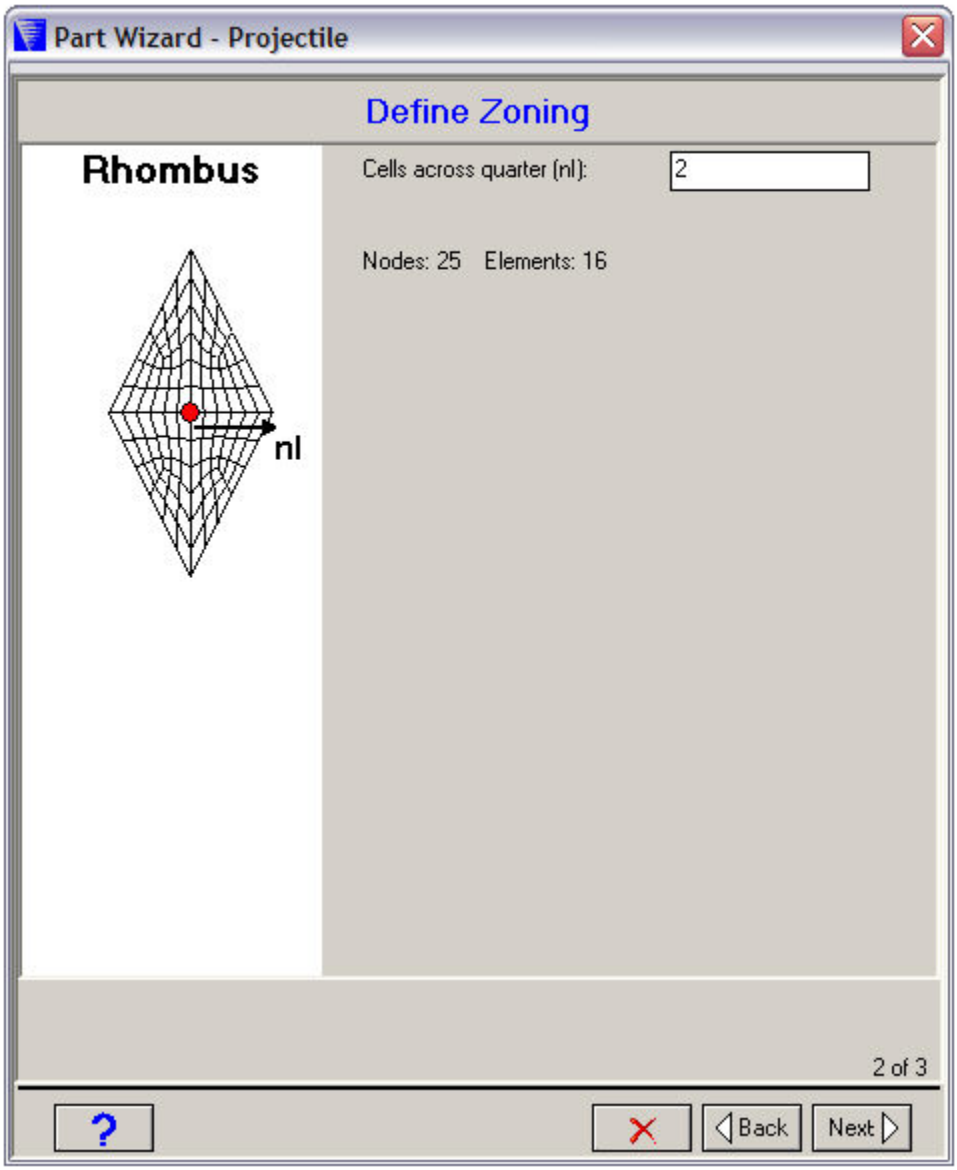
– **Increment**

The dimension you want to use for the fixed size cells.

– **Times**

The number of fixed size cells you want to use.

- **Rhombus**



Cells across quarter

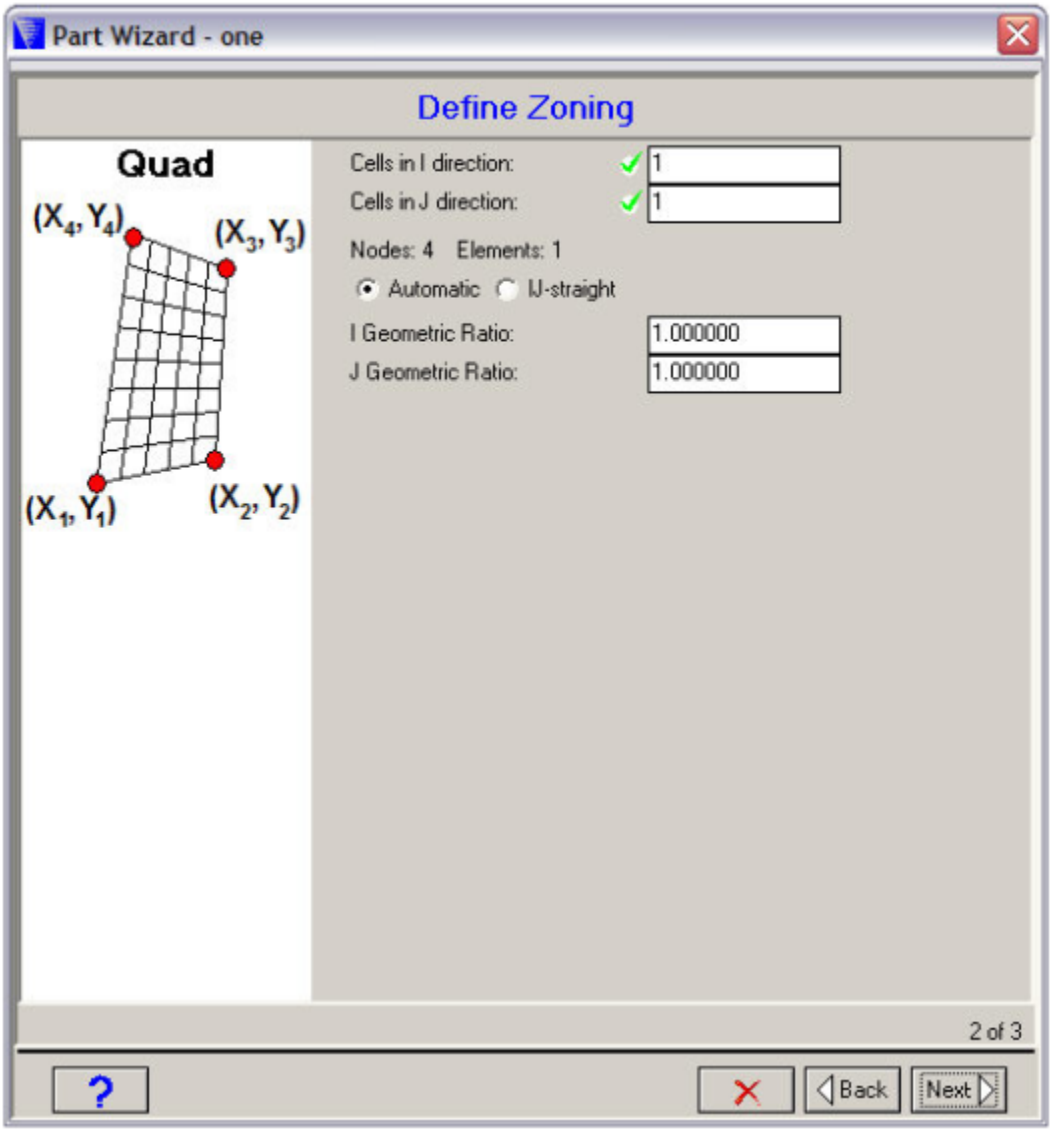
The number of cells (elements) you want to generate across a quarter of the rhombus.

Number of elements

Displayed below the input field is the index space that will be used for your Part and the number of nodes and elements required.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- **Quad**



Cells in I/J direction

The number of cells (elements) you want to use in the I and J directions. The corresponding number of nodes and elements required for your part will be displayed below.

Automatic

Select this option if you want the internal nodes to be spaced equipotentially.

IJ-Straight

Select this option if you want the nodes along each I and J line to be equally spaced along a straight line.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

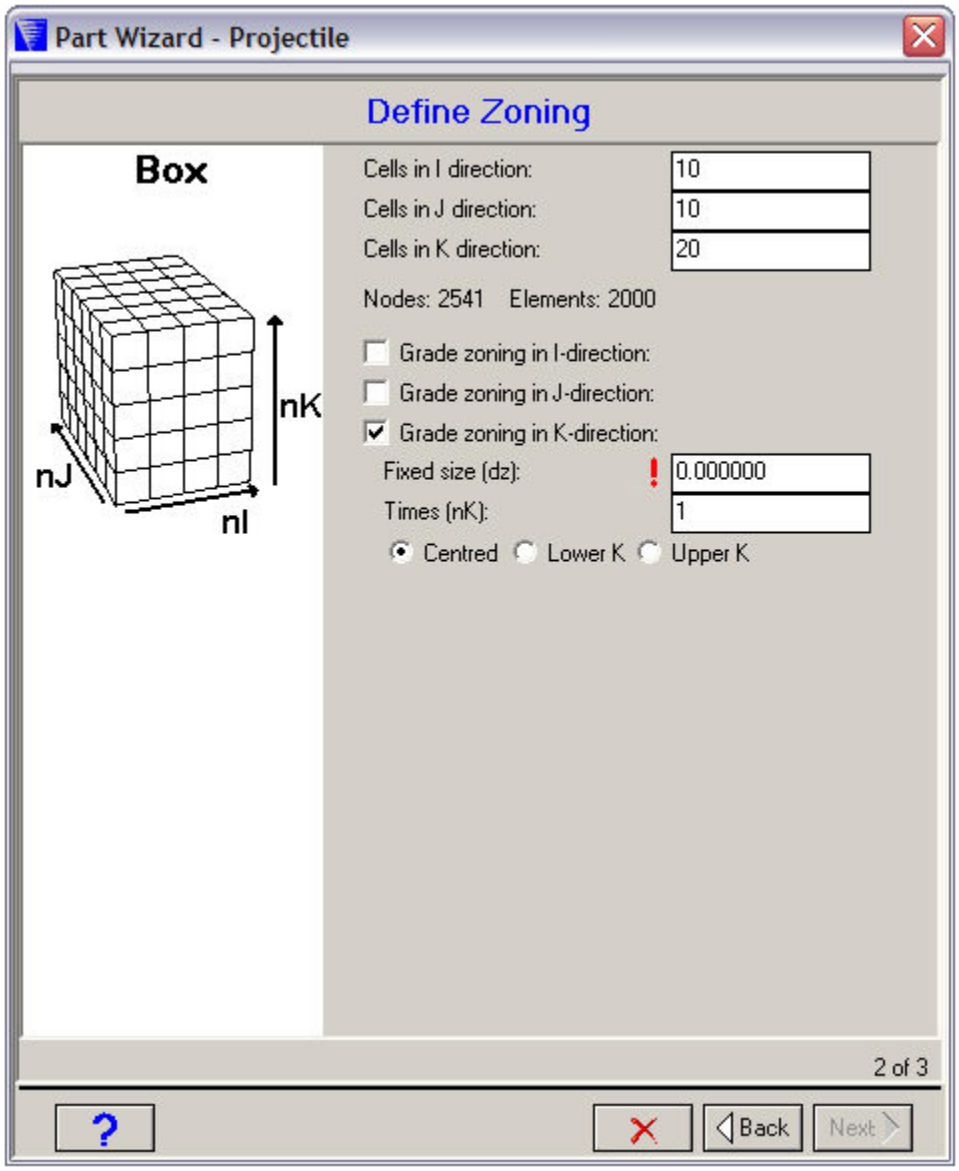
Geometric Ratios

Input geometric ratios in the fields provided if you want to grade the zoning instead of generating equally spaced nodes.

10.1.5. Parts - New - Define Zoning (3D)

This window lets you Define the zoning for your 3D Part.

- **Box**



Cells

The number of cells (elements) you want to use in the I, J and K directions. The corresponding number of nodes and elements required for your part will be displayed below.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Grade zoning

Check the boxes if you want to grade your zoning in any direction.

If you choose this option you can specify a range of cells which have a fixed size. The remaining cells will be smoothly graded to fit the overall dimension of the box you specified in the previous window.

– **Fixed size**

The dimension you want to use for the fixed size cells.

– **Times**

The number of fixed size cells you want to use.

– **Position**

Specify where you want to place the fixed size cells relative to the full I/J/K range.

Centered

At the center of the full I/J/K range.

Lower

At the lower end of the I/J/K range.

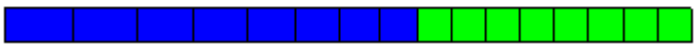
Upper

At the upper end of the I/J/K range.

e.g. ten fixed size cells centered :

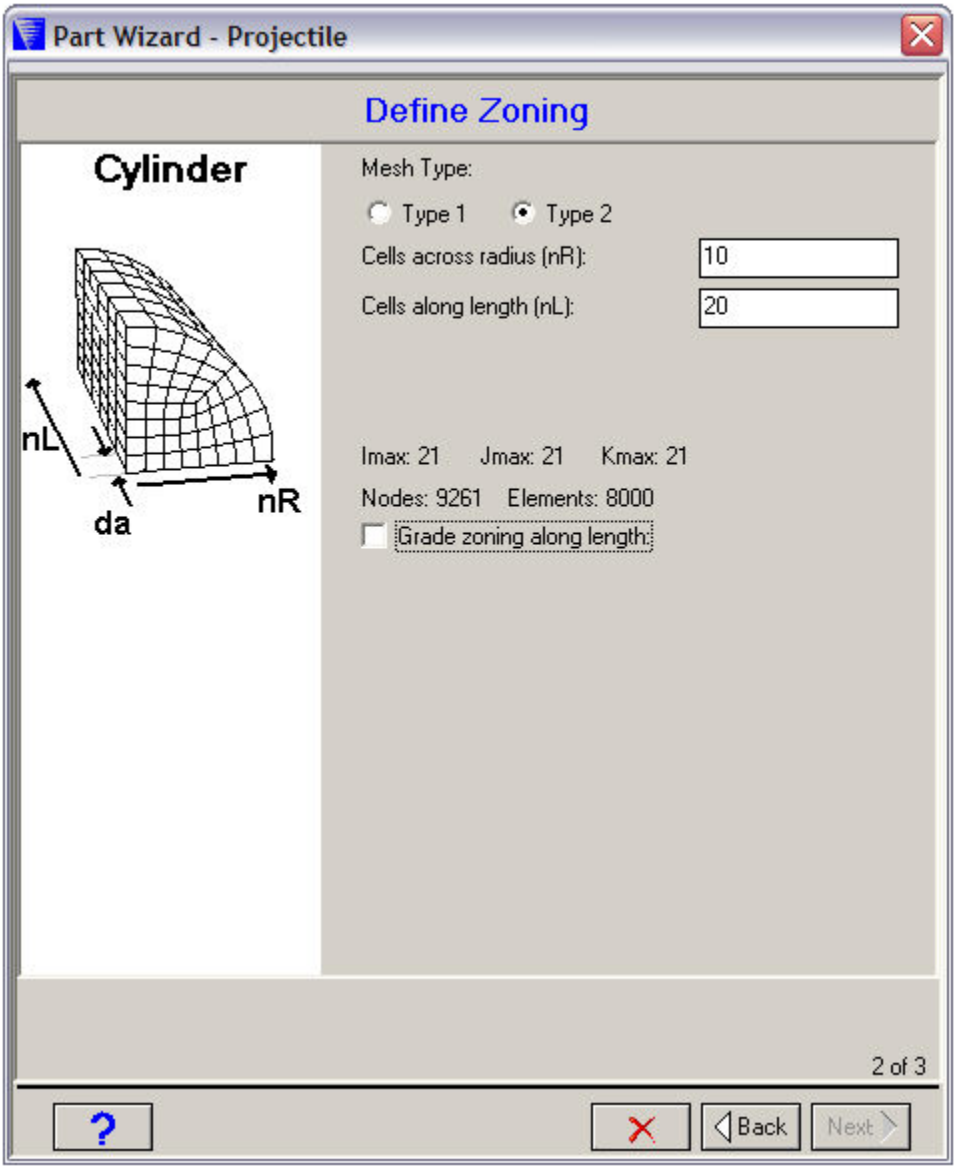
e.g. twelve fixed size cells at the lower end :

e.g. eight fixed size cells at the upper end :



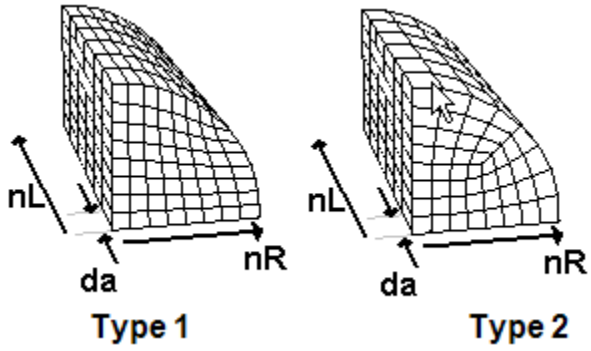
- **Cylinder**

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Mesh Type

Select a Type 1 or Type 2 mesh for your cylinder.



Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

The type 1 mesh is more efficient in using elements, but some elements are not very rectangular (which can reduce the accuracy of your solution).

All the elements of a type 2 mesh are reasonably rectangular, but about a quarter of the elements generated for this mesh are unused.

Cells across radius

The number of cells (elements) you want to use across the radius of your cylinder.

Cells along length

The number of cells (elements) you want to use along the length of your cylinder.

Number of elements

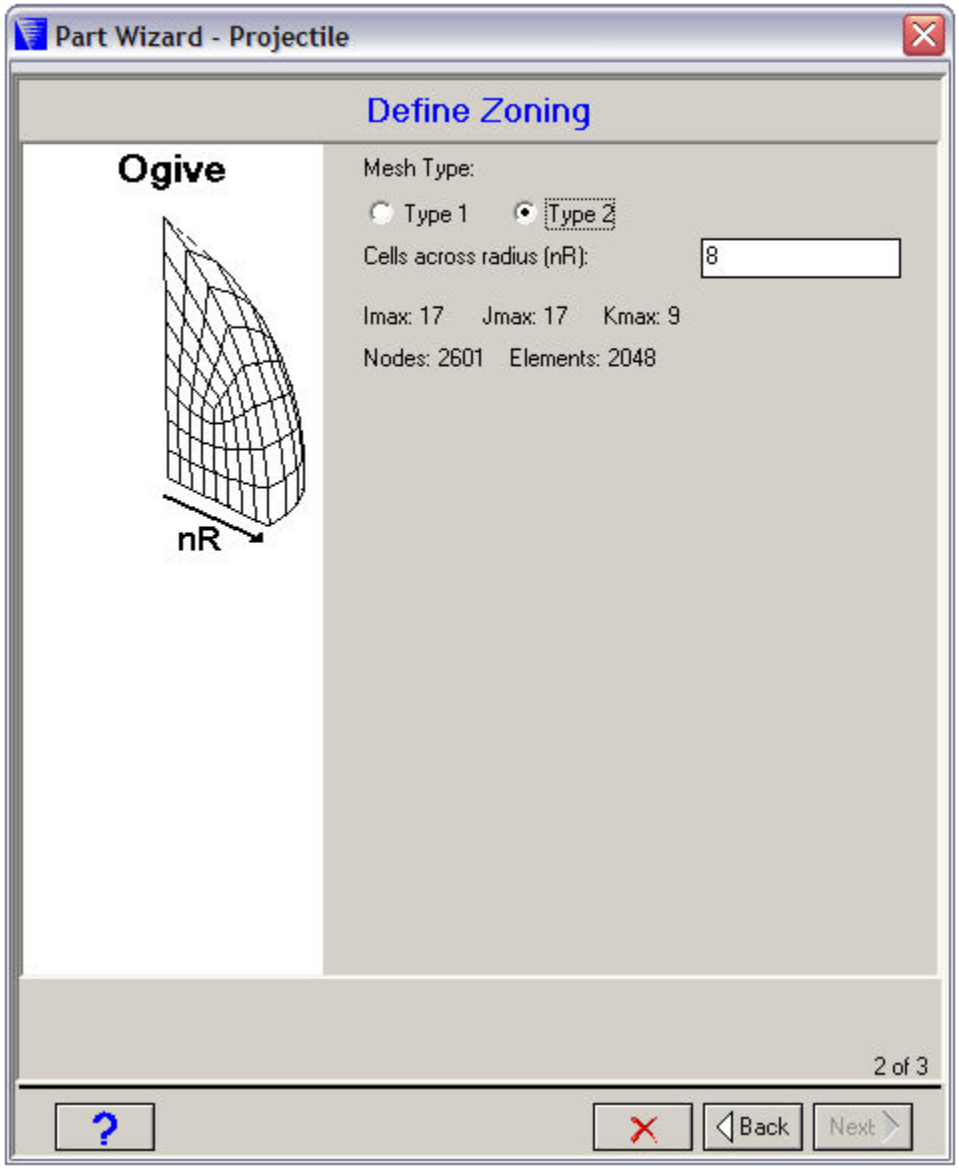
Displayed below the above input fields are the index space that will be used for your Part and the number of nodes and elements required.

Grade zoning along length

Check this box if you want to grade the zoning along the length of your cylinder (you will be asked to specify a **starting increment** to use).

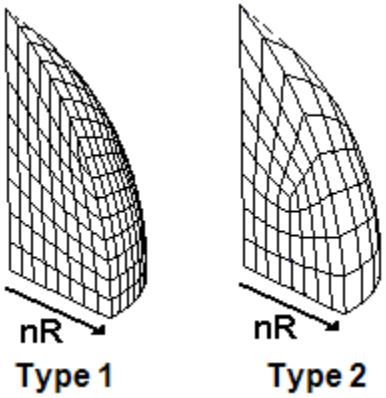
- **Ogive**

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Mesh Type

Select a Type 1 or Type 2 mesh for your ogive.



Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

The type 1 mesh is more efficient in using elements, but some elements are not very rectangular (which can reduce the accuracy of your solution).

All the elements of a type 2 mesh are reasonably rectangular, but some of the elements generated for this mesh are unused.

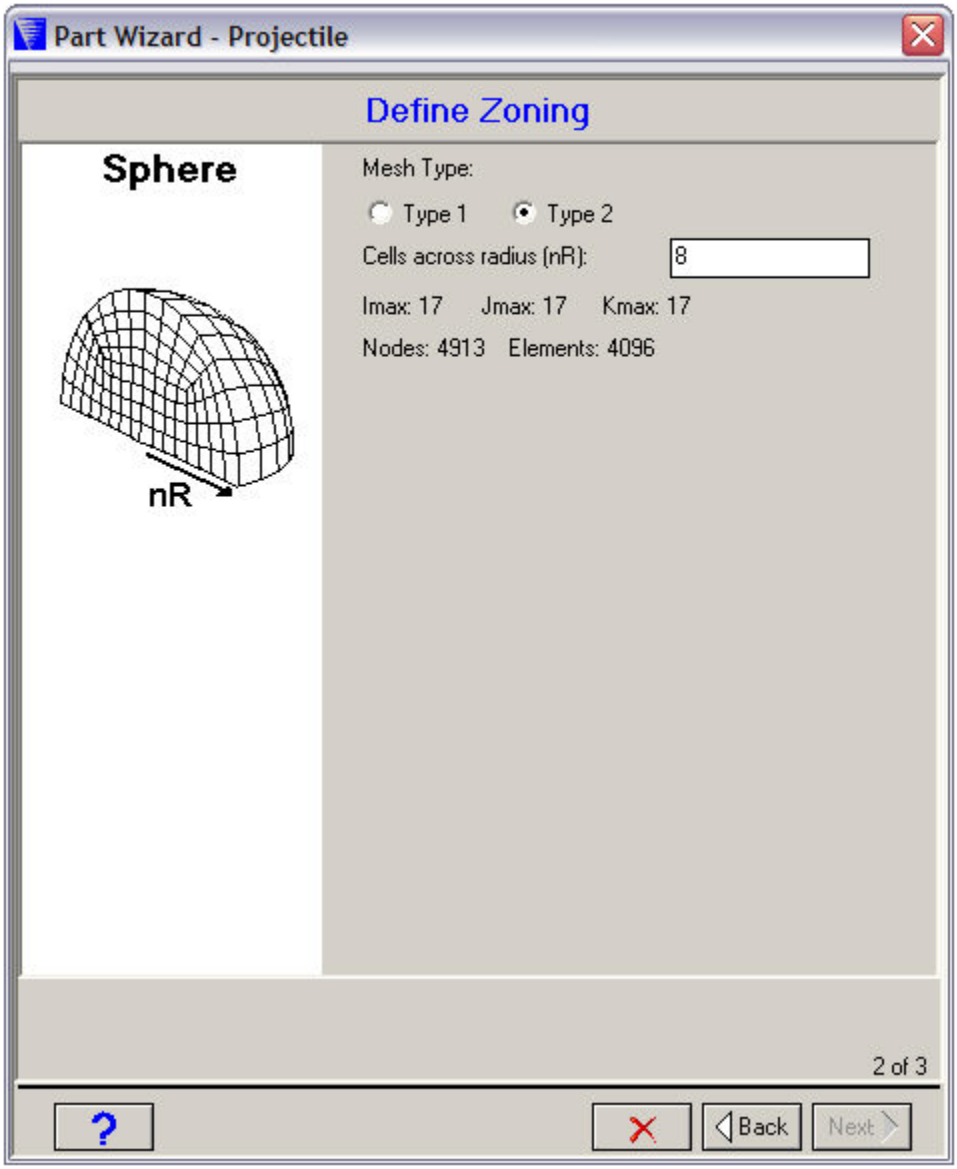
Cells across radius

The number of cells (elements) you want to use across the radius of your ogive.

Number of elements

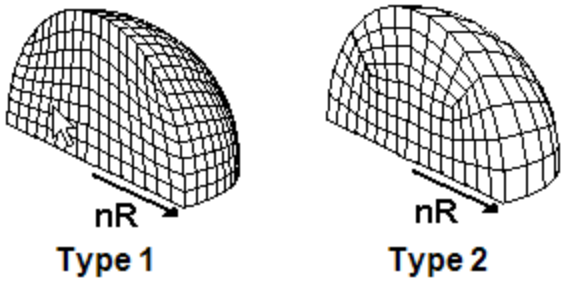
Displayed below the input fields is the index space that will be used for your Part and the number of nodes and elements required.

- **Sphere**



Mesh Type

Select a Type 1 or Type 2 mesh for your sphere.



The type 1 mesh is more efficient in using elements, but some elements are not very rectangular (which can reduce the accuracy of your solution).

All the elements of a type 2 mesh are reasonably rectangular, but some of the elements generated for this mesh are unused.

Cells across radius

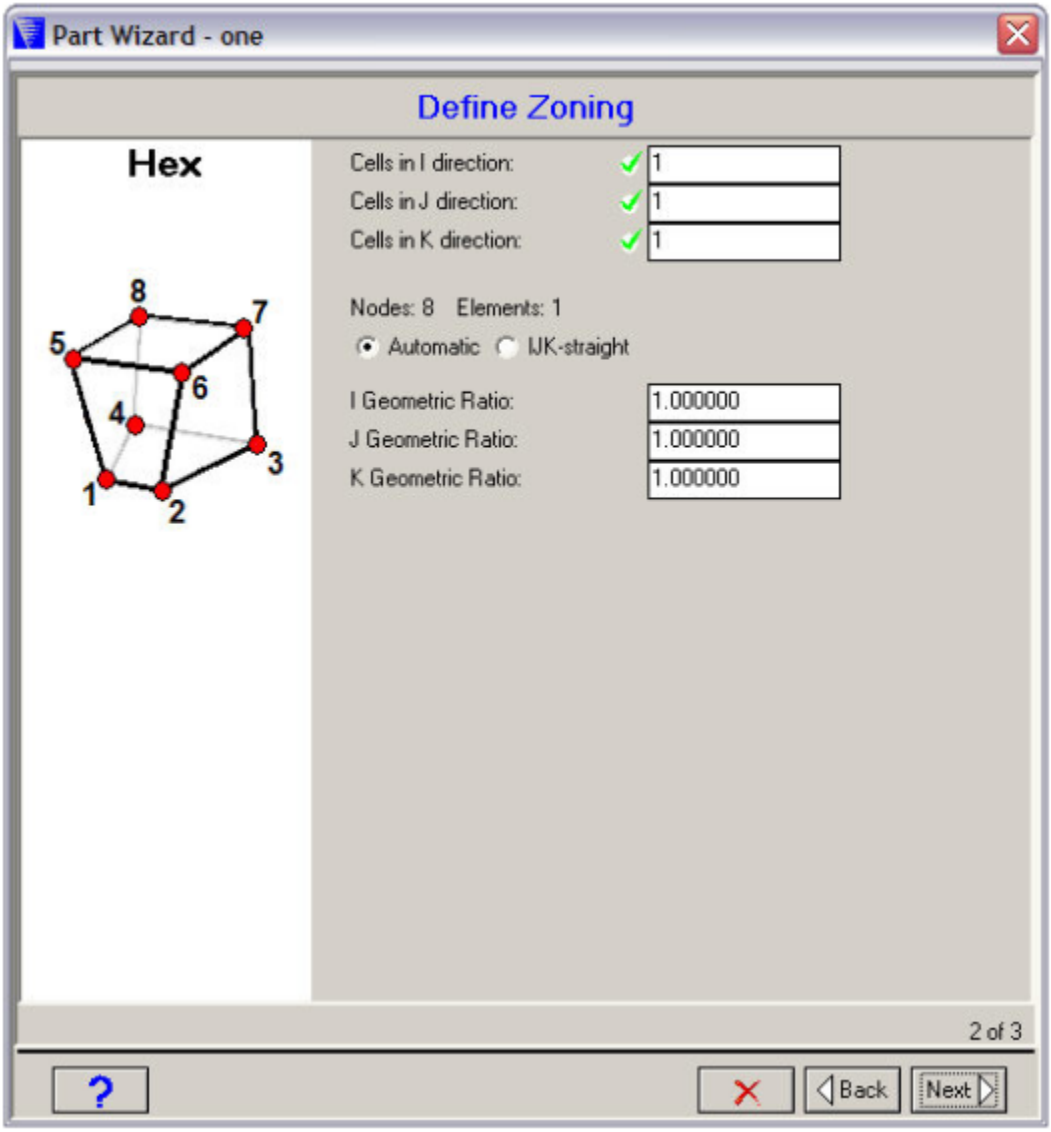
The number of cells (elements) you want to use across the radius of your sphere. For solid spheres, there is a maximum limit of 40 cells across the radius. For finer meshes than this, it is recommended that you use the Meshing application, where a hex mesh can be obtained by using the multi-zone and inflation layer operations. For more information see Autodyn Walkthrough Using Meshing (p. 519) in this guide.

Number of elements

Displayed below the input fields is the index space that will be used for your Part and the number of nodes and elements required.

- **Hex**

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Cells in I/J/K direction

The number of cells (elements) you want to use in the I, J, and K directions. The corresponding number of nodes and elements required for your part will be displayed below.

Automatic

Spaces the internal nodes equipotentially.

IJK-Straight

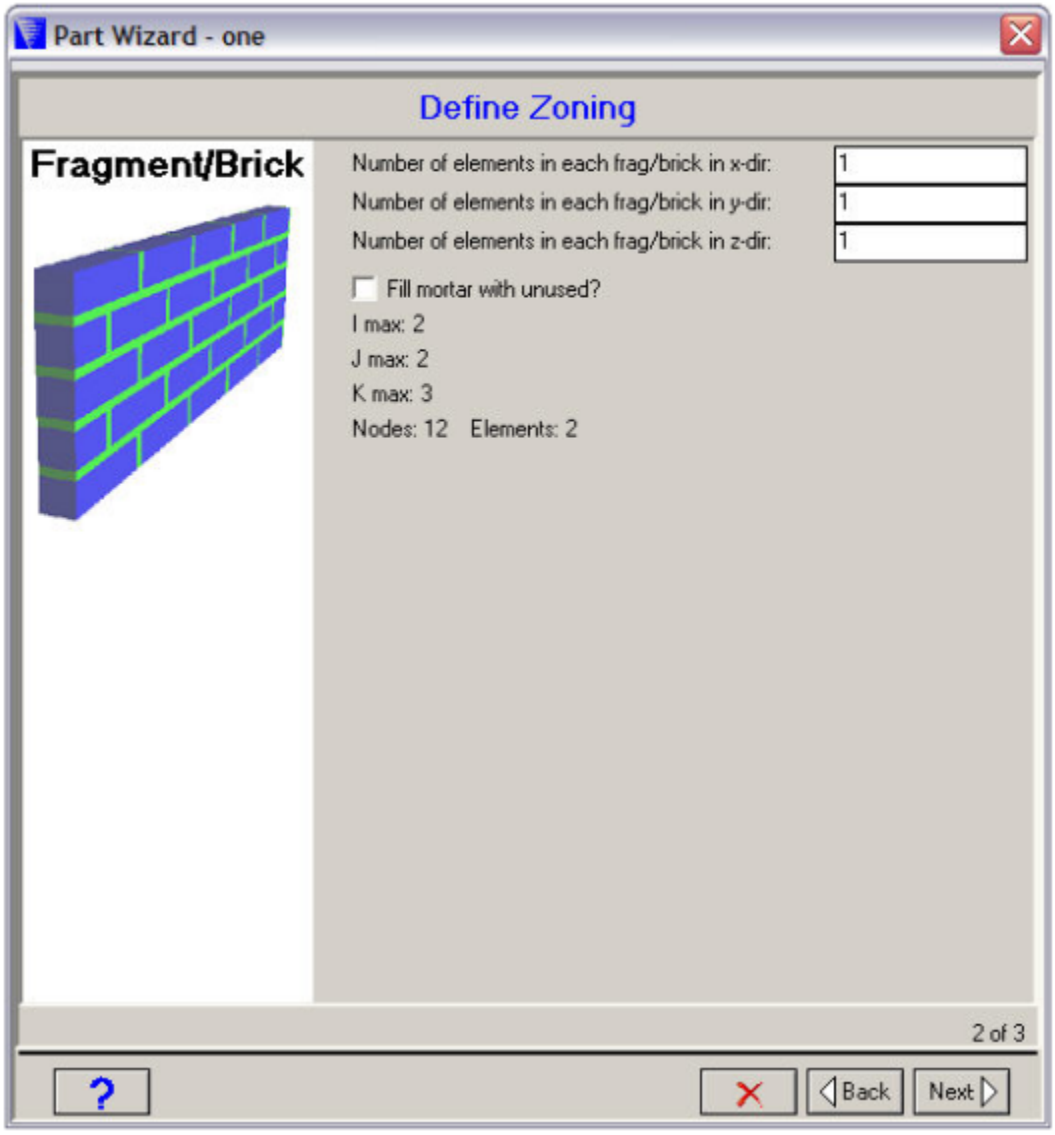
Spaces the nodes in a straight line equally along each I, J, and K line.

Geometric Ratios

Input geometric ratios in the fields provided if you want to grade the zoning instead of generating equally spaced nodes.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- **Frag/Brick**



Number of elements in each fragment/brick

Use these fields to set the number of elements per fragment/brick in the X, Y, and Z directions.

Fill mortar with unused?

Check this box if you do not want the mortar to be represented as a material.

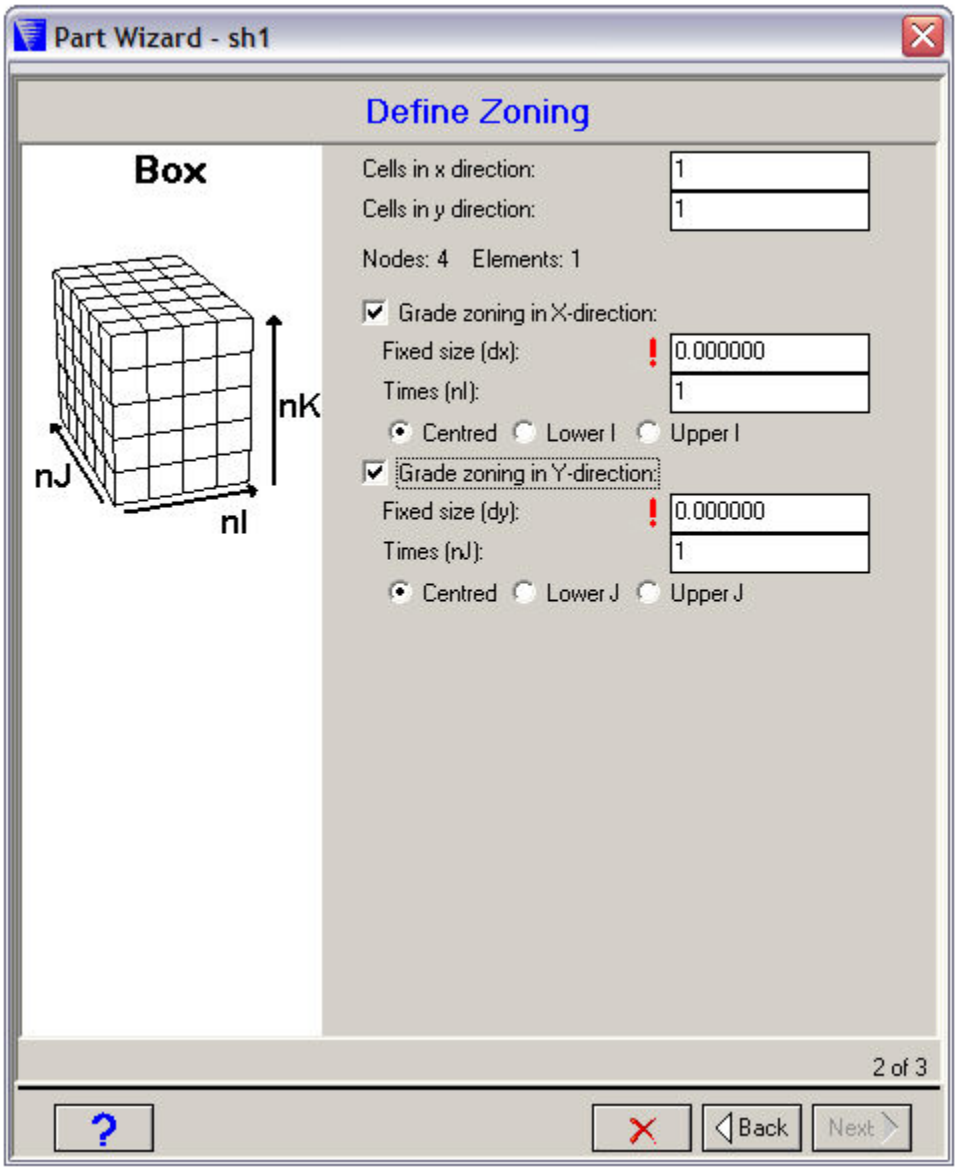
- If you check this box, no elements will be generated for the mortar.
- If you do not check this box, elements will be generated for the mortar. (If the mortar thickness is small compared to the dimension of the brick elements, the time step is likely to be reduced significantly.)

10.1.6. Parts - New - Define Zoning (3D Shell)

This window lets you Define the zoning for your Shell Part.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- **Plane**



Cells

The number of cells (elements) you want to use in the X, Y, Z directions. The corresponding number of nodes and elements required for your part will be displayed below.

Grade zoning

Check the boxes if you want to grade your zoning in any direction.

If you choose this option you can specify a range of cells which have a fixed size. The remaining cells will be smoothly graded to fit the overall dimension of the box you specified in the previous window.

Fixed size

The dimension you want to use for the fixed size cells.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Times

The number of fixed size cells you want to use.

Position

Specify where you want to place the fixed size cells relative to the full I/J/K range.

Centered

At the center of the full I/J/K range.

Lower

At the lower end of the I/J/K range.

Upper

At the upper end of the I/J/K range.

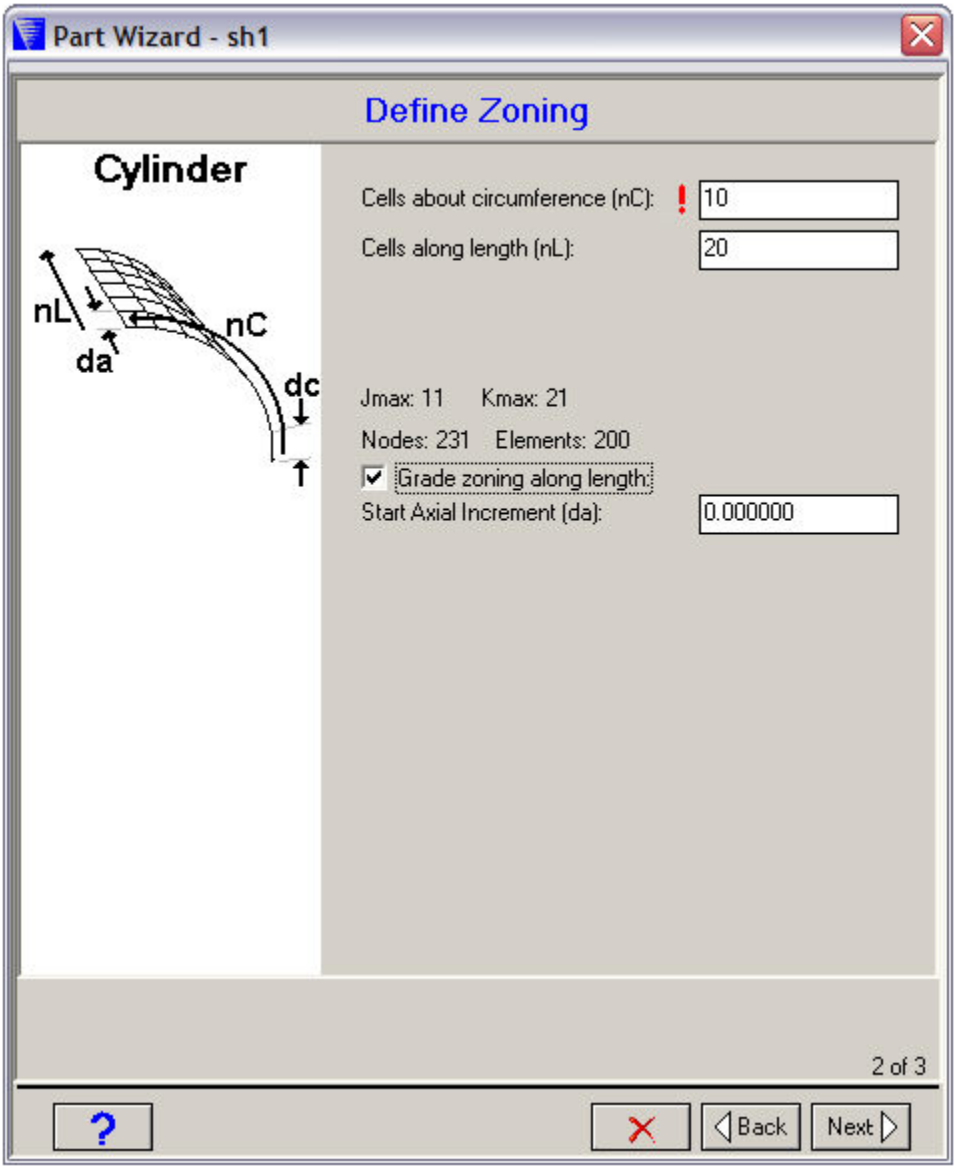
e.g. ten fixed size cells centered :

e.g. twelve fixed size cells at the lower end :

e.g. eight fixed size cells at the upper end :

- **Cylinder**

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Cells about circumference

The number of cells (elements) you want to use about the circumference of your cylinder.

Cells along length

The number of cells (elements) you want to use along the length of your cylinder.

Number of elements

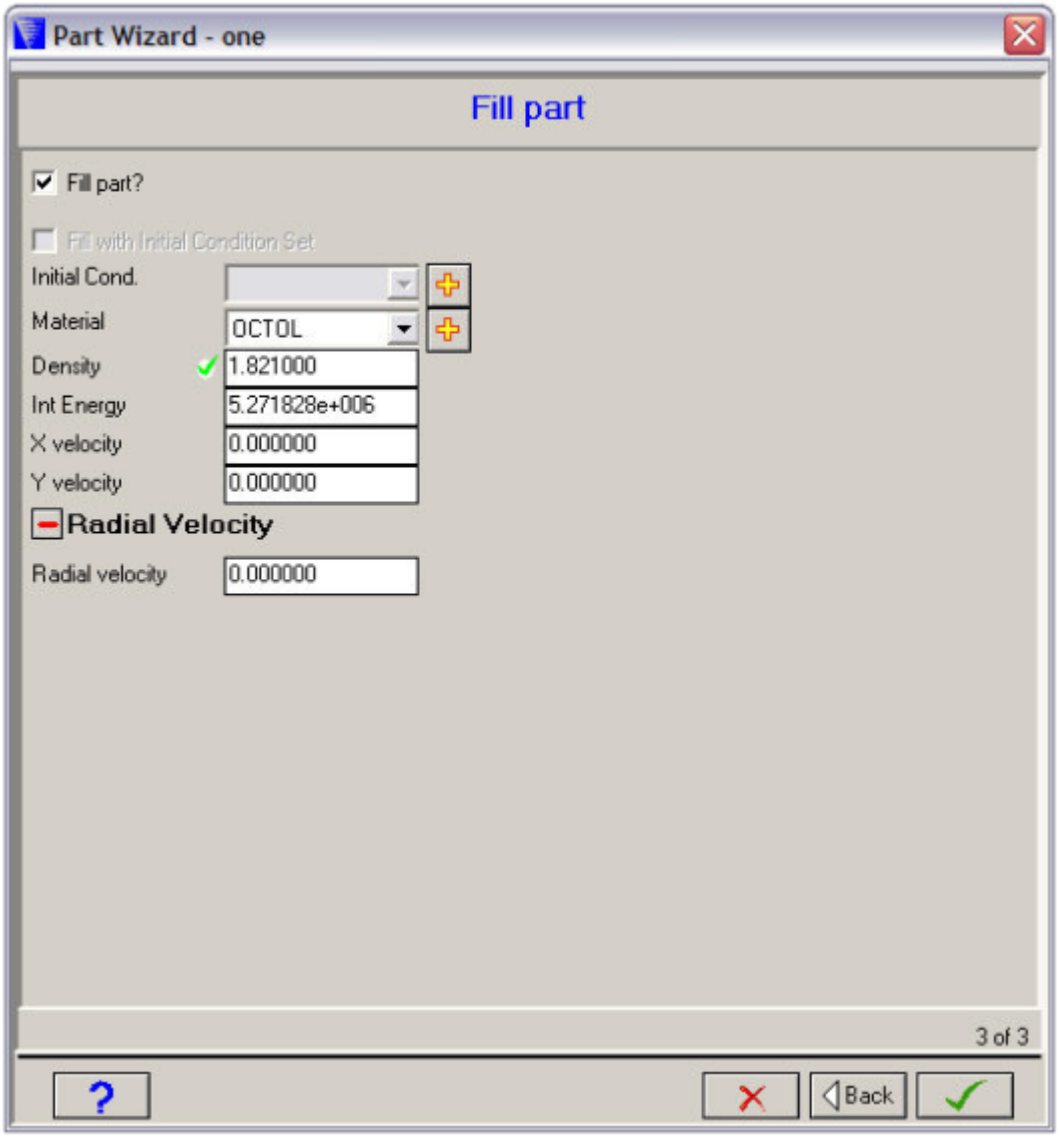
Displayed below the above two input fields is the index space that will be used for your Part and the number of nodes and elements required.

Grade zoning along length

Check this box if you want to grade the zoning along the length of your cylinder (you will be asked to specify a **starting increment** to use).

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

10.1.7. Parts - New - Fill Part (2D)



This window lets you fill the part you are creating.

Fill Part

Check this box if you want the Part Wizard to fill your Part.

The Part Wizard lets you fill your entire Part with a single material and initial values.

If you want to fill your part with multiple materials or initial conditions, this can be done by clicking the **Fill** button after completing the Part Wizard.

Fill with Initial Condition Set

Check this box if you want to fill your Part using an Initial Condition Set.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

The advantage of filling a Part using an Initial Condition Set is that if you later change one or more parameters in the set, you do not have to refill your Part.

Use the pull-down menu to select the Initial Condition Set you want to use.

You can define new Initial Condition Sets by clicking the **Init.Cond.** button on the Navigation Bar or by clicking next to the option in this window.

Material

If you do not want to use an Initial Condition Set, select the material you want to use to fill your Part from this Pull-down menu.

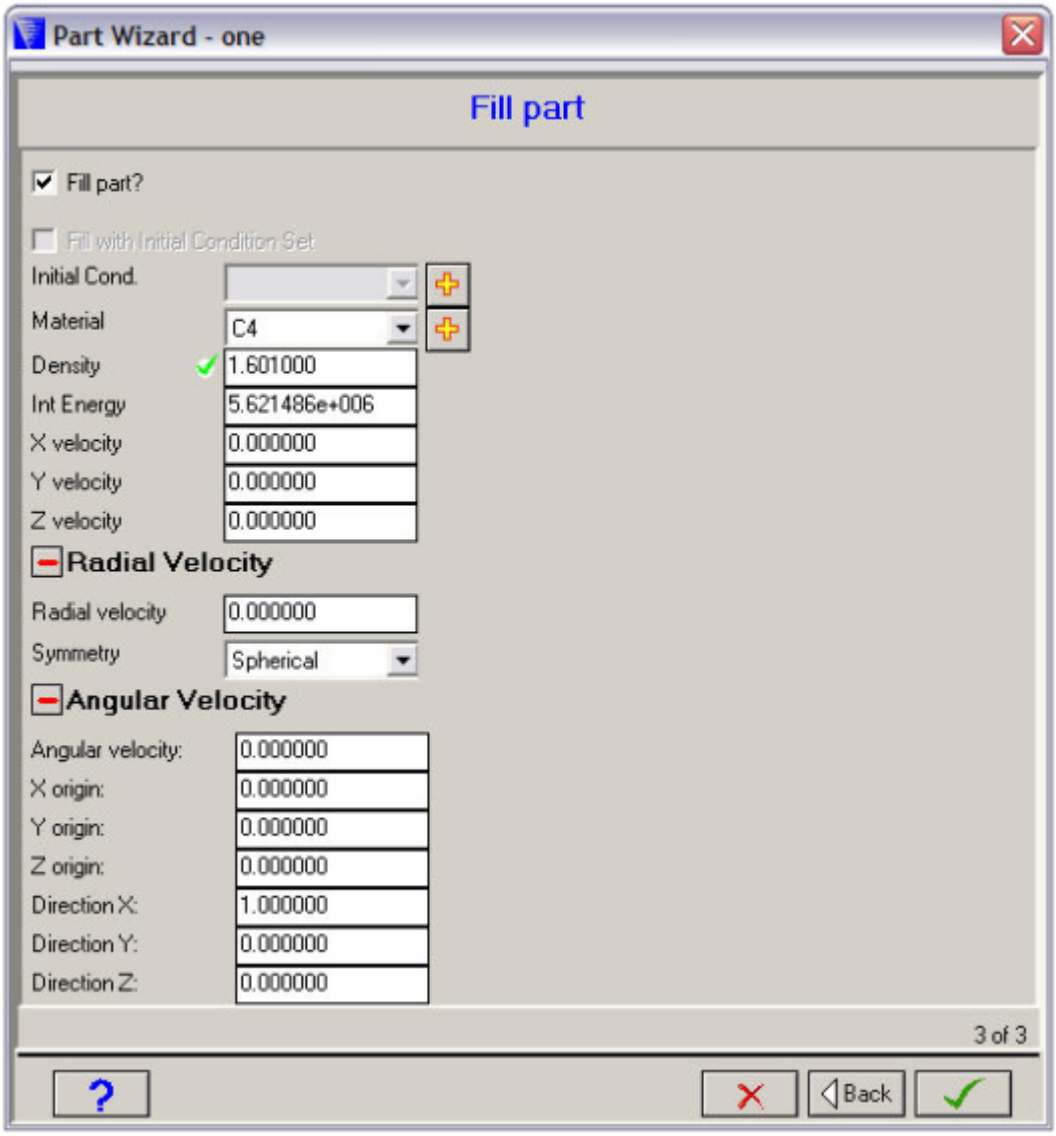
You can define new materials by clicking the **Materials** button on the Navigation Bar or by clicking next to the option in this window.

Parameters

If you do not want to use an Initial Condition Set, enter parameter values for your fill in the fields provided (**Density**, **Energy**, **Velocity**).

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

10.1.8. Parts - New - Fill Part (3D)



This window lets you fill the part you are creating.

Fill Part

Check this box if you want the Part Wizard to fill your Part.

The Part Wizard lets you fill your entire Part with a single material and initial values.

If you want to fill your part with multiple materials or initial conditions, this can be done by clicking the **Fill** button after completing the Part Wizard.

Fill with Initial Condition Set

Check this box if you want to fill your Part using an Initial Condition Set.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

The advantage of filling a Part using an Initial Condition Set is that if you later change one or more parameters in the set, you do not have to refill your Part.

Use the pull-down menu to select the Initial Condition Set you want to use.

You can define new Initial Condition Sets by clicking the **Init.Cond.** button on the Navigation Bar or by clicking next to the option in this window.

Material

If you do not want to use an Initial Condition Set, select the material you want to use to fill your Part from this Pull-down menu.

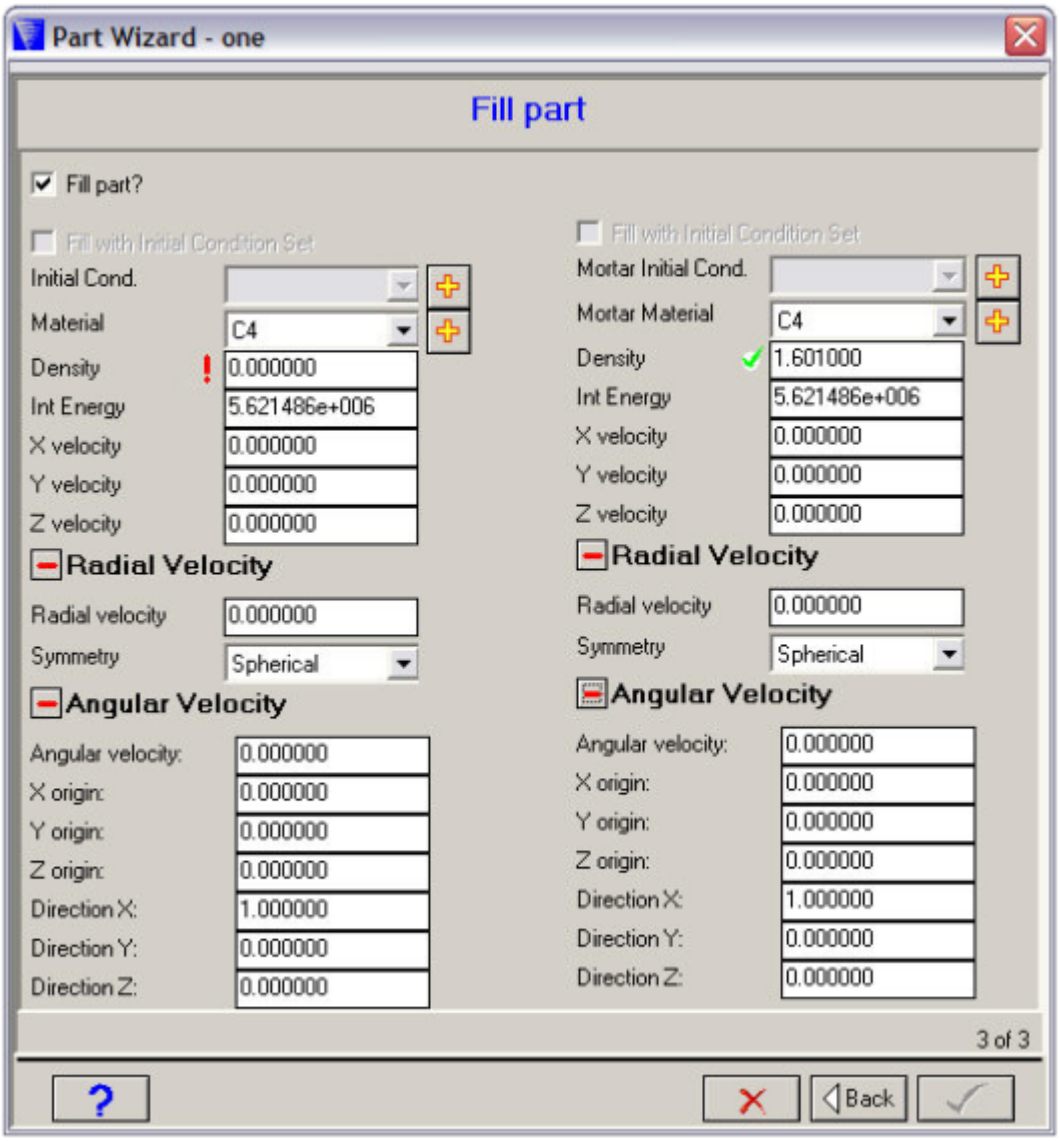
You can define new materials by clicking the **Materials** button on the Navigation Bar or by clicking next to the option in this window.

Parameters

If you do not want to use an Initial Condition Set, enter parameter values for your fill in the fields provided (**Density**, **Energy**, **Velocity**).

Frag/Brick

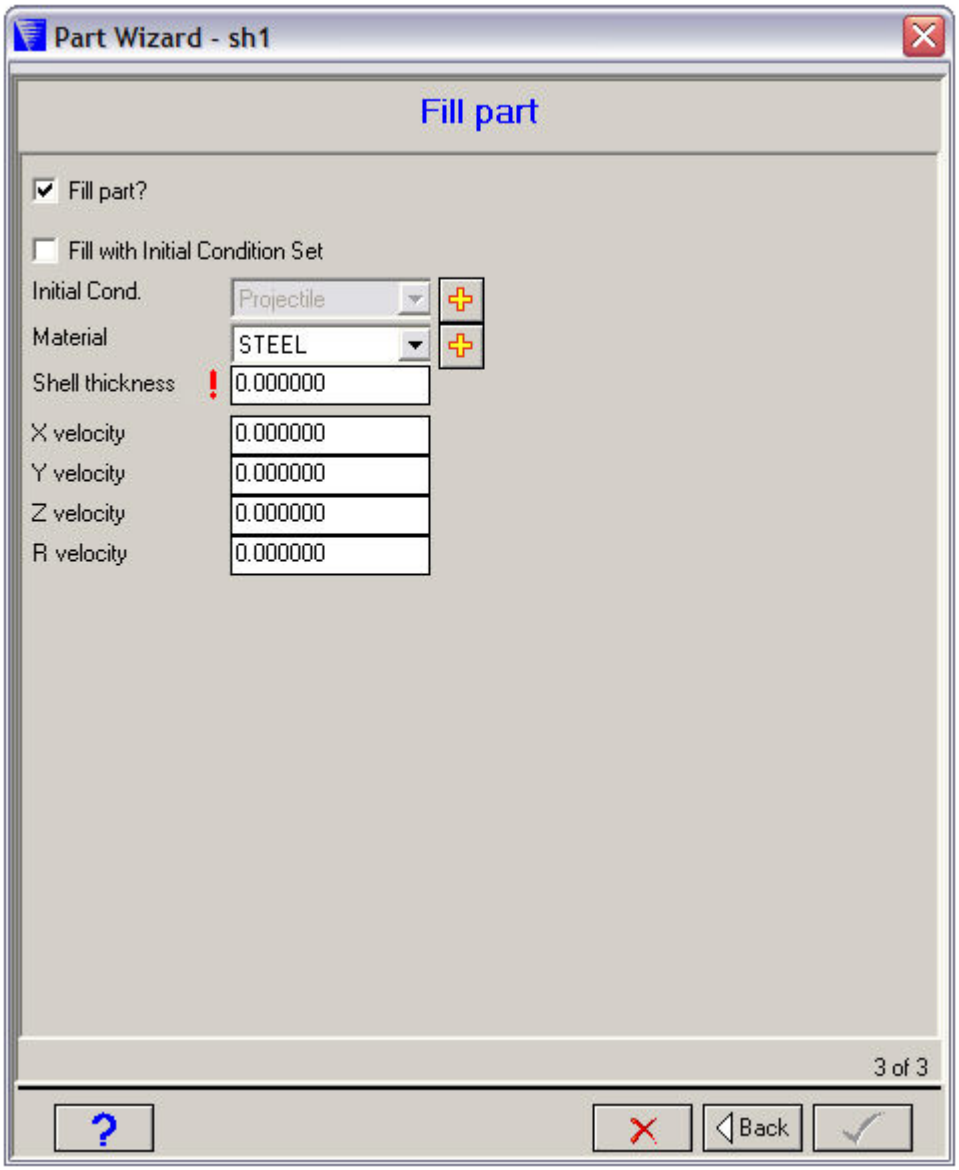
Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Enter the material data for the brick and mortar materials.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

10.1.9. Parts - New - Fill Part (3D Shell)



This window lets you fill the shell part you are creating.

Fill Part

Check this box if you want the Part Wizard to fill your Part.

The Part Wizard lets you fill your entire Part with a single material and initial values.

If you want to fill your part with multiple materials or initial conditions, this can be done by clicking the **Fill** button after completing the Part Wizard.

Fill with Initial Condition Set

Check this box if you want to fill your Part using an Initial Condition Set.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

The advantage of filling a Part using an Initial Condition Set is that if you later change one or more parameters in the set, you do not have to refill your Part.

Use the pull-down menu to select the Initial Condition Set you want to use.

You can define new Initial Condition Sets by clicking the **Init.Cond.** button on the Navigation Bar or by clicking next to the option in this window.

Material

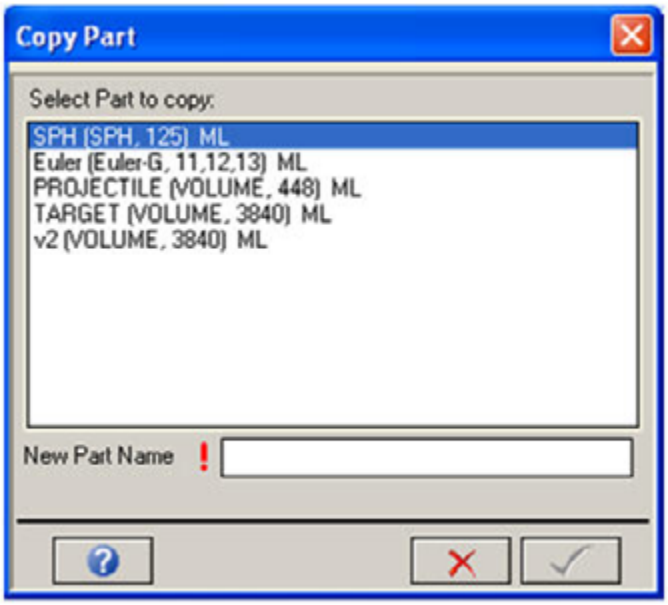
If you do not want to use an Initial Condition Set, select the material you want to use to fill your Part from this Pull-down menu.

You can define new materials by clicking the **Materials** button on the Navigation Bar or by clicking next to the option in this window.

Parameters

If you do not want to use an Initial Condition Set, enter parameter values for your fill in the fields provided (**Density**, **Energy**, **Velocity**).

10.2. Parts - Copy



This window lets you copy an existing Part to a new Part.

Part List

Lists all the existing Parts in your model. Select the part you want to copy from this list.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

New Part Name

The name of your new Part.

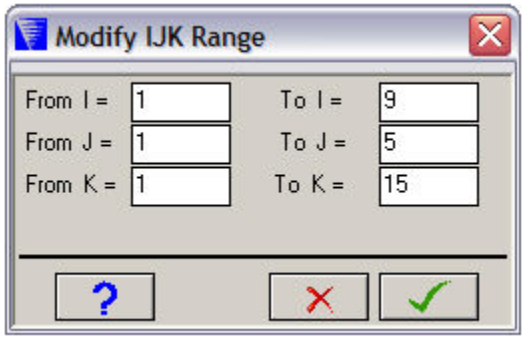
Clicking creates your new part.

10.3. Parts - Delete

This window lets you delete Parts from your model.

Select the Part(s) you want to delete and click .

10.4. Parts - IJK Range



This window lets you modify the IJK range of a structured Part.

Enter new IJK ranges for the Part in the fields provided.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

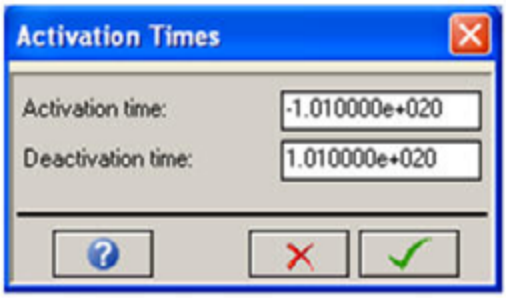
If you cut a range, the corresponding elements are deleted from the Part.

If you extend a range, new (unused) elements are added to the Part.

Use zero or negative indices to add I, J or K lines to the lower end of ranges.

For example, changing the I-range above to run From I = -4 To I = 12, will add 5 I-lines at the lower end of the Part and 3 I-lines at the upper end of the Part. The new I-range will then run from 1 to 17 (with the original I-lines moved to I=5 through I=14)

10.5. Parts - Activation



This window lets you set the activation and deactivation times for a structured Part. Activation / Deactivation times cannot be set for unstructured Parts. The default values ensure the Part is active for the entire calculation.

Activation Time

The activation time for the Part. The Part will not be computed until this activation time is reached.

Deactivation Time

The deactivation time for the Part. The Part will not be computed after this deactivation time is reached.

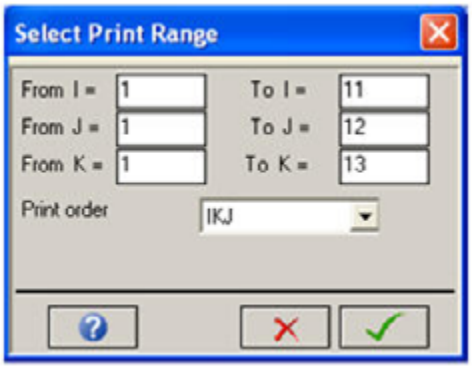
10.6. Parts - Rename

This window lets you rename a Part.

Type the new name for the Part in the field provided.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

10.7. Parts - Print



This window lets you set the index range of a structured Part you want to use for grid data written to the print file (**.prt**) for this Part.

You can set the frequency at which this data is written (by cycle or times) in the **Output** panel (click **Output** on the Navigation Bar).

Index Range

The index range for print data for this Part.

Print order

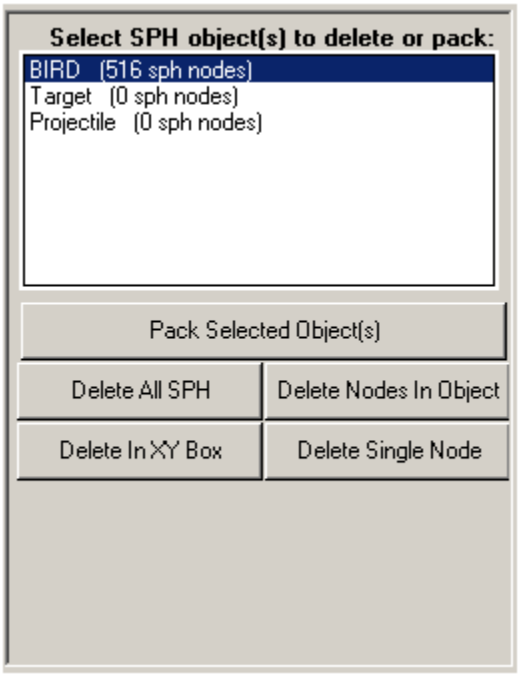
Use this pull-down menu to set the order in which data will be printed.

For example, IKJ means J will vary faster than K and K will vary faster than I.

10.8. Parts - Pack (Fill)

When SPH parts are selected in the parts menu, clicking the "Pack (Fill)" button displays the following panel in the lower region of the parts menu.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



This panel allows you to fill SPH objects with SPH nodes.

Pack Selected Object(s)

Pack the selected objects with SPH nodes. You will be prompted for the fill material, the SPH node size and velocity to use for the packing operation.

Delete All SPH

Delete all SPH nodes in your model.

Delete Nodes in Object

Delete the SPH nodes in the selected SPH objects.

Delete in XY Box

Delete the SPH nodes in a box defined by the XYZ coordinates of the origin and the size of the box in the X, Y, and Z directions.

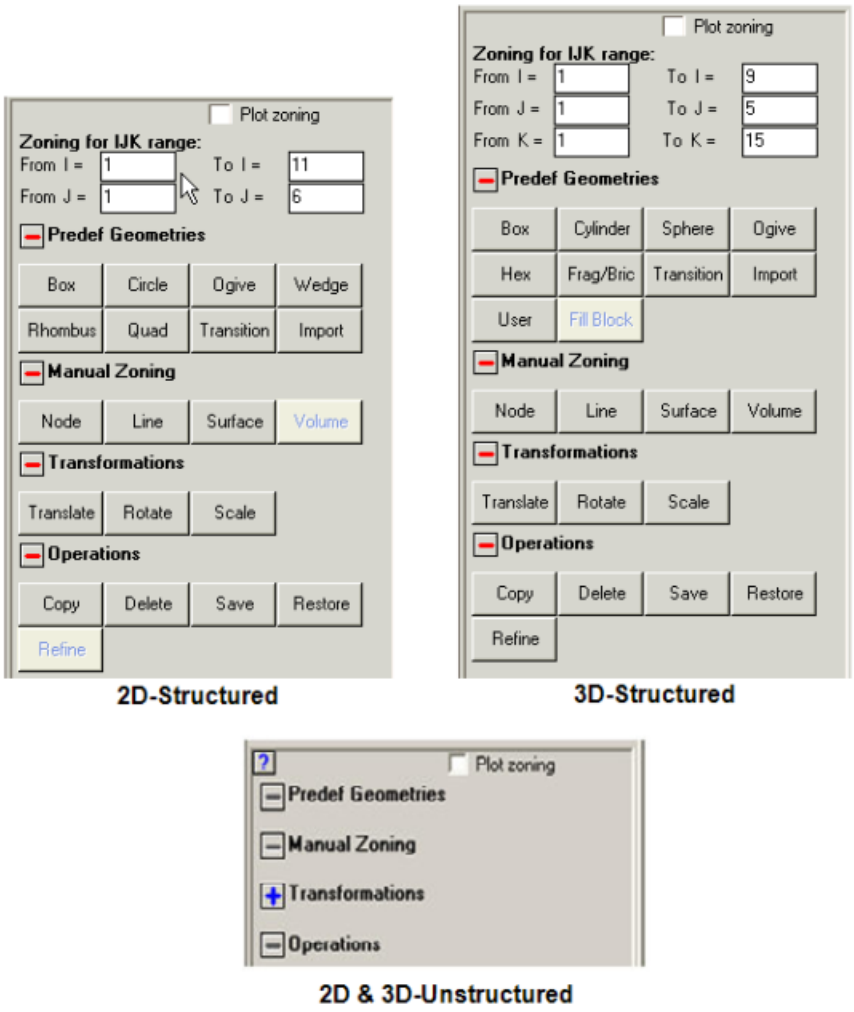
Delete Single Node

Delete a single SPH node, by specifying its local IJK index. You can use the Examine option to interactively select SPH nodes, and determine their index.

10.9. Parts - Zoning

Clicking the **Zoning** button in the **Parts** panel displays the following dialog in the lower region of the panel :

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



This dialog lets you define the zoning (grid coordinates) for the selected Part.

For 2D and 3D unstructured Parts the geometry and meshing cannot be defined. The only option for unstructured Parts is the Transformation option, like translation, rotation or scaling.

Plot zoning

Standard plotting will only show elements that contain material, not unused elements.

Checking this box shows all elements, including unused elements.

Zoning for IJK range

These fields let you specify the IJK range you want zoning operations to act on. The default values are for the entire Part.

Predef Geometries

Box

Zone a box.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Cylinder

Zone a cylinder.

Sphere

Zone a sphere.

Ogive

Zone an ogive.

Frag/Bric

Zone a fragment / brick wall.

Transition

Zone a grid transition

Import

Import zoning from an external file (.zon)

User

Generate zoning from a user subroutine.

Fill Block

Zone and fill an Euler-FCT grid using data from an external file.

Manual Zoning

These options let you build a grid by specifying nodes and generating lines, surfaces and 3D volumes.

Click to reveal the following buttons :

Node

Define the coordinates of a node.

Line

Generate coordinates along a line in index space.

Surface

Generate the coordinates of an index plane.

Volume

Define the coordinates within the current IJK Range.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Transformations

These options let you translate, rotate and scale your zoning.

Click to reveal the following buttons :

Translate

Translate zoning.

Rotate

Rotate zoning.

Scale

Scale zoning.

Operations

These options let you perform various operations on your zoning.

Click to reveal the following buttons :

Copy

Copy zoning from an existing Part to the current Part.

Delete

Delete zoning.

Save

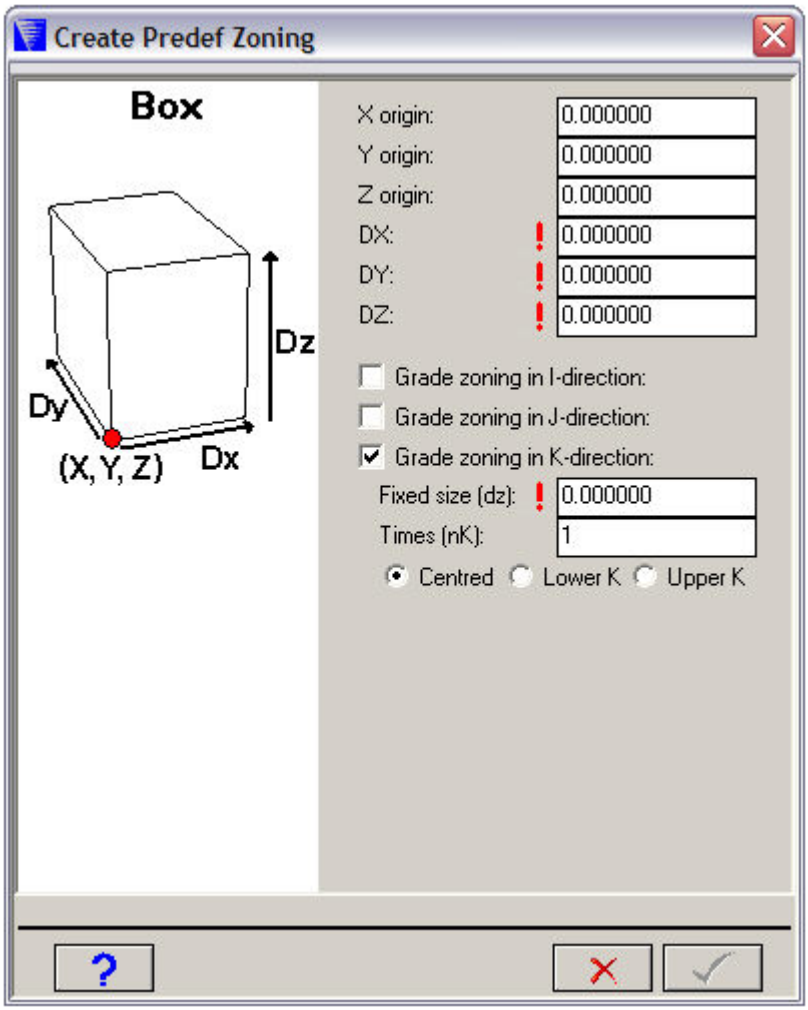
Temporarily save your zoning.

Restore

Restore zoning you have temporarily saved.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

10.9.1. Parts - Zoning - Box



This window lets you generate a box with its axes parallel to the X, Y, and Z axes.

Origin

The X, Y, and Z coordinates of the lower corner of the box.

Box Dimensions

The dimensions of the box (DX, DY, DZ)

Grade zoning

Check the boxes if you want to grade your zoning in any direction.

If you choose this option you can specify a range of cells which have a fixed size. The remaining cells will be smoothly graded to fit the overall dimension of the box you specified in the previous window.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Fixed size

The dimension you want to use for the fixed size cells.

Times

The number of fixed size cells you want to use.

Position

Specify where you want to place the fixed size cells relative to the full I/J/K range.

Centered

At the center of the full I/J/K range.

Lower

At the lower end of the I/J/K range.

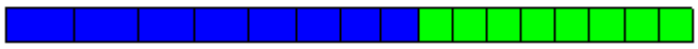
Upper

At the upper end of the I/J/K range.

e.g. ten fixed size cells centered :

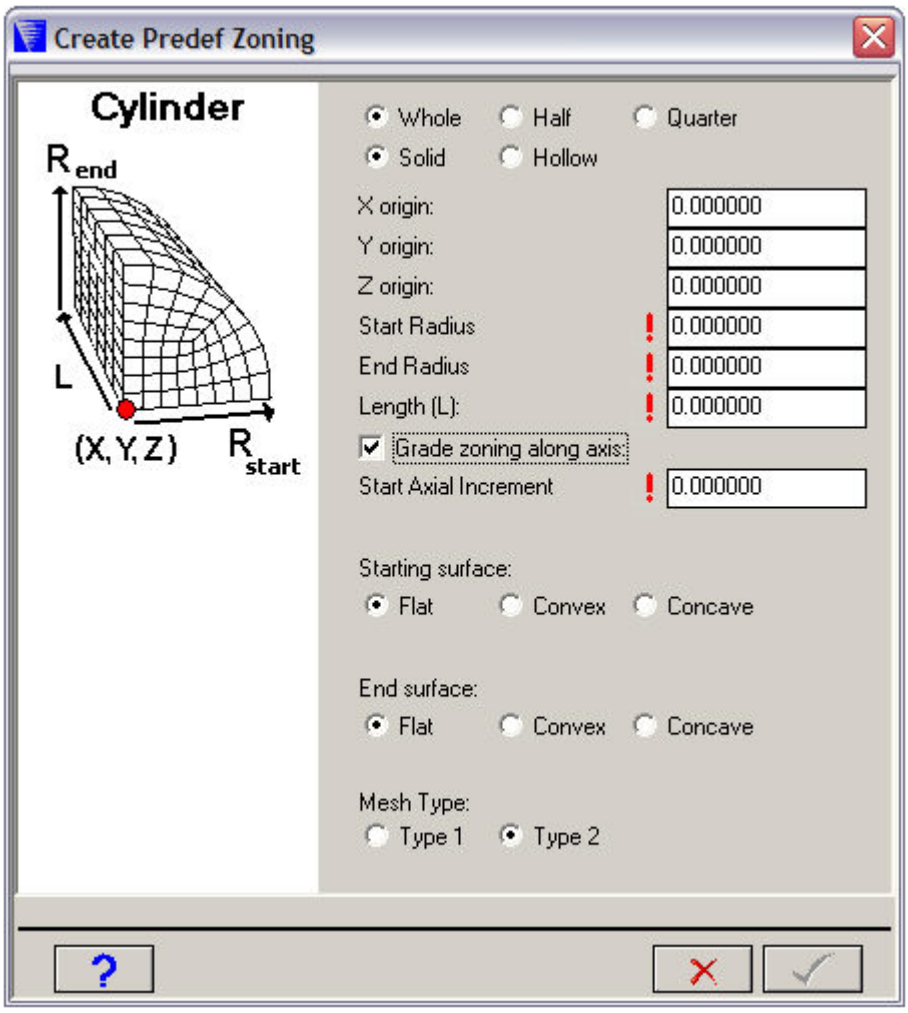
e.g. twelve fixed size cells at the lower end :

e.g. eight fixed size cells at the upper end :



Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

10.9.2. Parts - Zoning - Cylinder



This window lets you generate a cylinder with its axis parallel to the Z axis.

Section

The section you want to generate (Whole, Half or Quarter). This should match any symmetry planes you have defined for your model.

Solid or Hollow

Choose either a solid or hollow cylinder.

If you choose a hollow cylinder, you will have to specify inner and outer radii for the cylinder.

Origin

The X, Y, and Z coordinates for the center of the starting surface of your cylinder.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Start Radius / End Radius

You can specify different radii for the starting and end faces of your cylinder. For a regular cylinder (constant radius), set the starting radius equal to the end radius.

Length

The length of your cylinder.

Grade zoning along axis

Check this box if you want to grade the zoning along the length of your cylinder (you will asked to specify a geometric ratio to use).

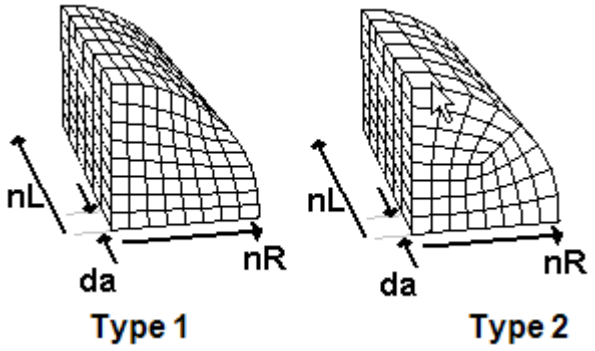
Start Surface / End surface

The starting and end surfaces of your cylinder can be flat, concave or convex.

If you select a concave or convex surface, you will have to specify its radius of curvature.

Mesh Type

Select a Type 1 or Type 2 mesh for your cylinder.

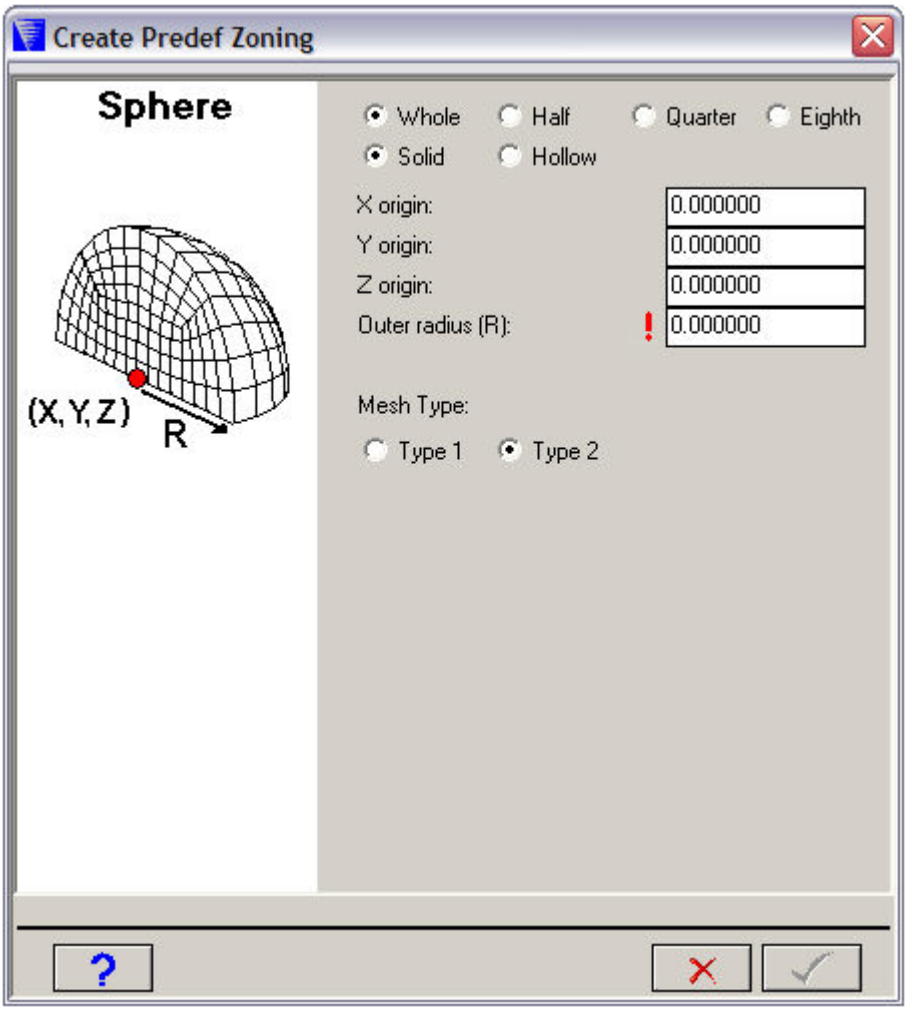


The type 1 mesh is more efficient in using elements, but some elements are not very rectangular (which can reduce the accuracy of your solution).

All the elements of a type 2 mesh are reasonably rectangular, but about a quarter of the elements generated for this mesh are unused.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

10.9.3. Parts - Zoning - Sphere



This window lets you generate a sphere.

Section

The section you want to generate (Whole, Half, Quarter or Eighth). This should match any symmetry planes you have defined for your model.

Solid or Hollow

Choose either a solid or hollow sphere.

If you choose a hollow sphere, you will have to specify inner and outer radii for the sphere.

Origin

The X, Y, and Z coordinates for the center of the sphere.

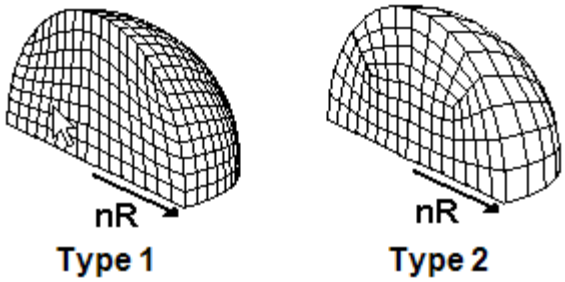
Radius

The radius of the sphere.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Mesh Type

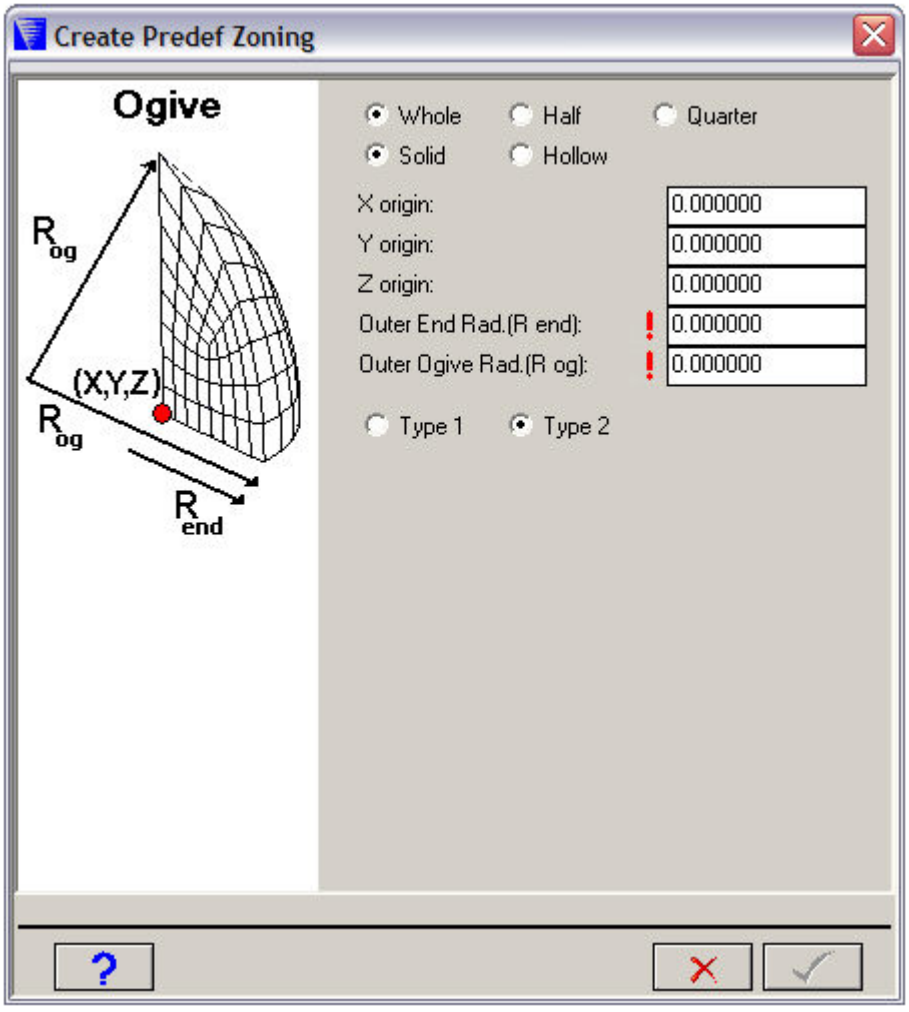
Select a Type 1 or Type 2 mesh for your sphere.



The type 1 mesh is more efficient in using elements, but some elements are not very rectangular (which can reduce the accuracy of your solution).

All the elements of a type 2 mesh are reasonably rectangular, but about a quarter of the elements generated for this mesh are unused.

10.9.4. Parts - Zoning - Ogive



Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

This window lets you generate an ogive with its axis parallel to the Z axis.

Section

The section you want to generate (Whole, Half or Quarter). This should match any symmetry planes you have defined for your model.

Solid or Hollow

Choose either a solid or hollow ogive.

If you choose a hollow ogive, you will have to specify inner and outer radii for the ogive.

Origin

The X, Y, and Z coordinates for the center of your ogive.

End Radius

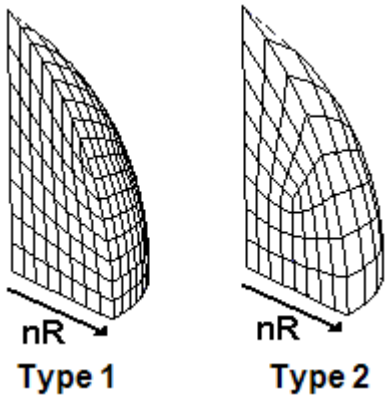
The end radius of your ogive.

Ogive Radius

The ogive radius.

Mesh Type

Select a Type 1 or Type 2 mesh for your ogive.

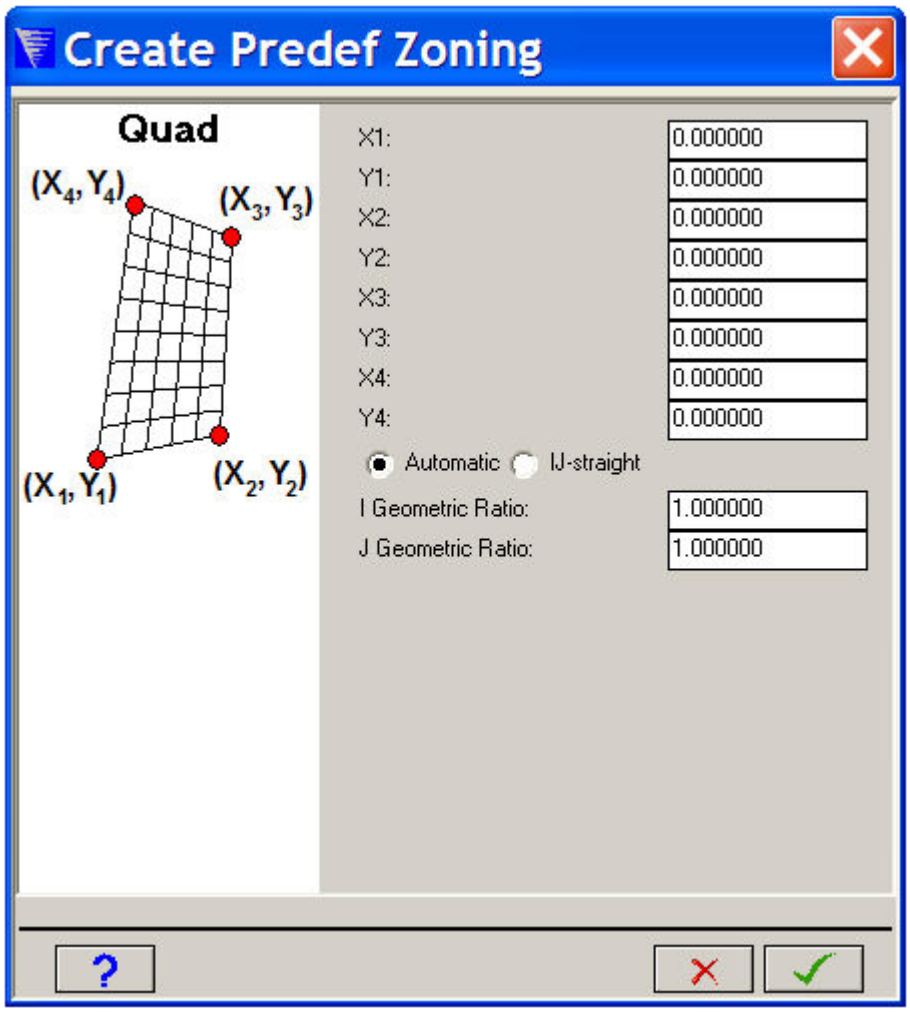


The type 1 mesh is more efficient in using elements, but some elements are not very rectangular (which can reduce the accuracy of your solution).

All the elements of a type 2 mesh are reasonably rectangular, but about a quarter of the elements generated for this mesh are unused.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

10.9.5. Parts - Zoning - Quad



Coordinates X1, Y1

The first point of the quad.

Coordinates X2, Y2

The second point of the quad.

Coordinates X3, Y3

The third point of the quad.

Coordinates X4, Y4

The fourth point of the quad.

Cells in I/J direction

The number of cells (elements) you want to use in the I and J directions. The corresponding number of nodes and elements required for your part will be displayed below.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Automatic

Select this option if you want the internal nodes to be spaced equipotentially.

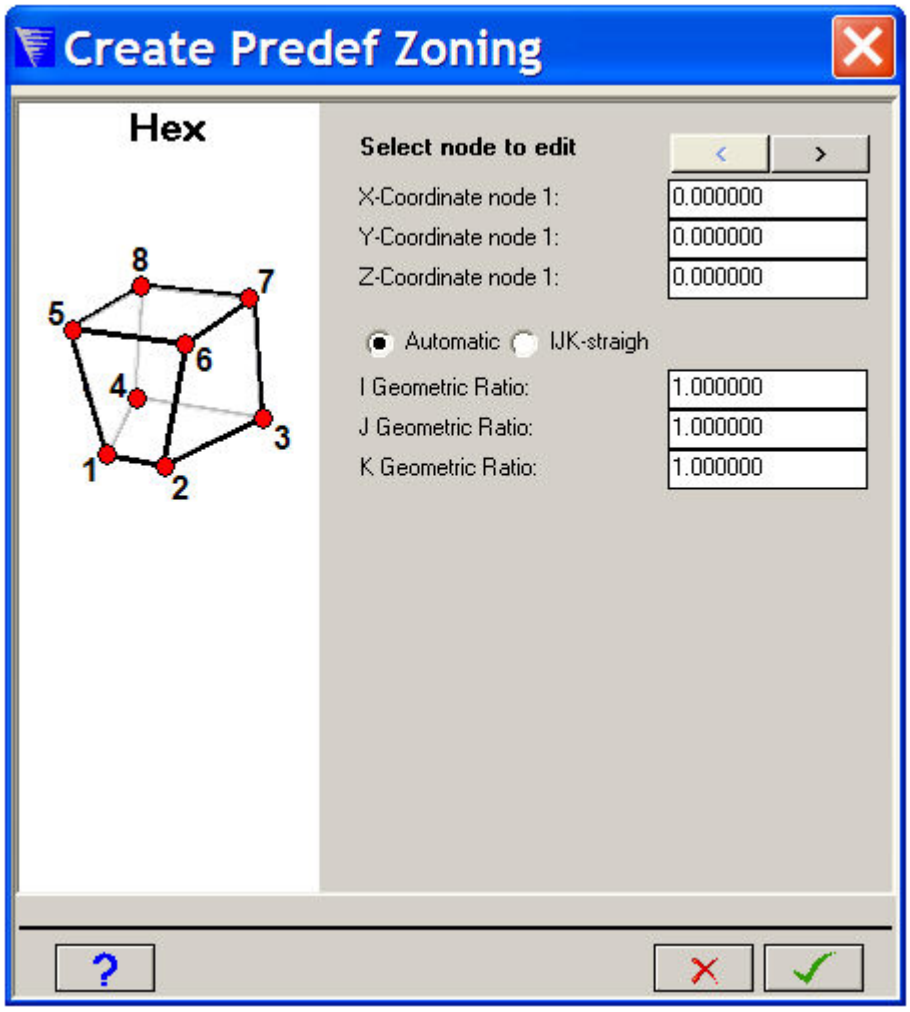
IJ-Straight

Select this option if you want the nodes along each I and J line to be equally spaced along a straight line

Geometric Ratios

Input geometric ratios in the fields provided if you want to grade the zoning instead of generating equally spaced nodes.

10.9.6. Parts - Zoning - Hex



Select Node to Edit

Use the arrow buttons to select each of the 8 nodes in turn.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Coordinates

The X, Y, and Z coordinates of the node.

Cells in I/J/K direction

The number of cells (elements) you want to use in the I, J and K directions. The corresponding number of nodes and elements required for your part will be displayed below.

Automatic

Select this option if you want the internal nodes to be spaced equipotentially.

IJK-Straight

Select this option if you want the nodes along each I, J and K line to be equally spaced along a straight line.

Geometric Ratios

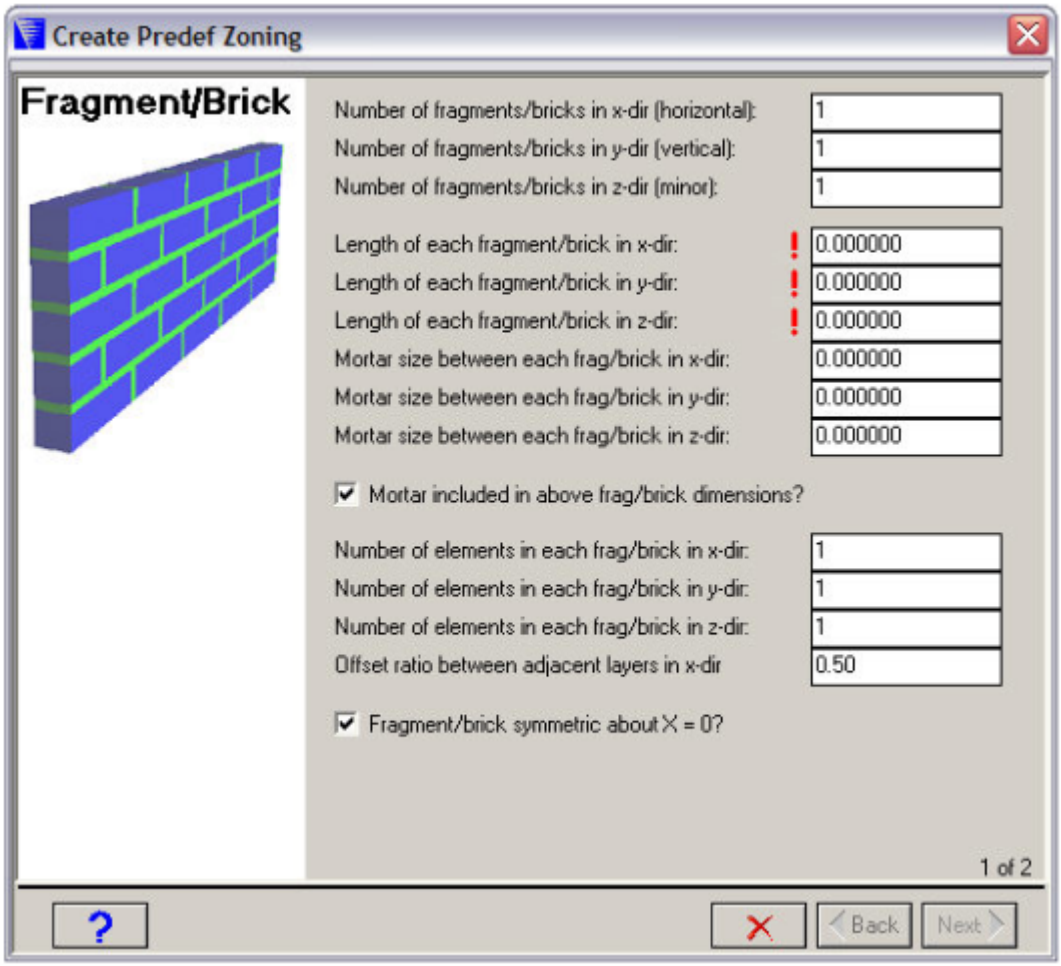
Input geometric ratios in the fields provided if you want to grade the zoning instead of generating equally spaced nodes.

10.9.7. Parts - Zoning - Frag/Bric (Page 1)

Two dialog windows let you build a fragment or brick wall.

This first window lets you generate the zoning.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



The fragment/brick wall is built with its axes parallel to the X, Y, and Z axes.

The X direction runs the length of the wall.

The Y direction runs the height of the wall.

The Z direction runs through the thickness of the wall.

Number of fragments/bricks

The number of fragments/bricks in the X, Y, and Z directions.

Length of each fragment/brick

The length of a single brick/fragment in the X, Y, and Z directions.

Mortar size between each fragment/brick

The mortar size (or gap) between fragments/bricks.

Mortar included in dimension?

Check this box if you want the mortar size to be included in the length of each brick/fragment.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Number of elements in each fragment/brick

The number of elements per fragment/brick in the X, Y, and Z directions.

Offset ratio

The offset ratio for the fragments/bricks.

This is how the bricks are staggered along the length of the wall. For example, if the offset is 0.5 (the default), the bricks will be staggered halfway across each other as shown in the above figure.

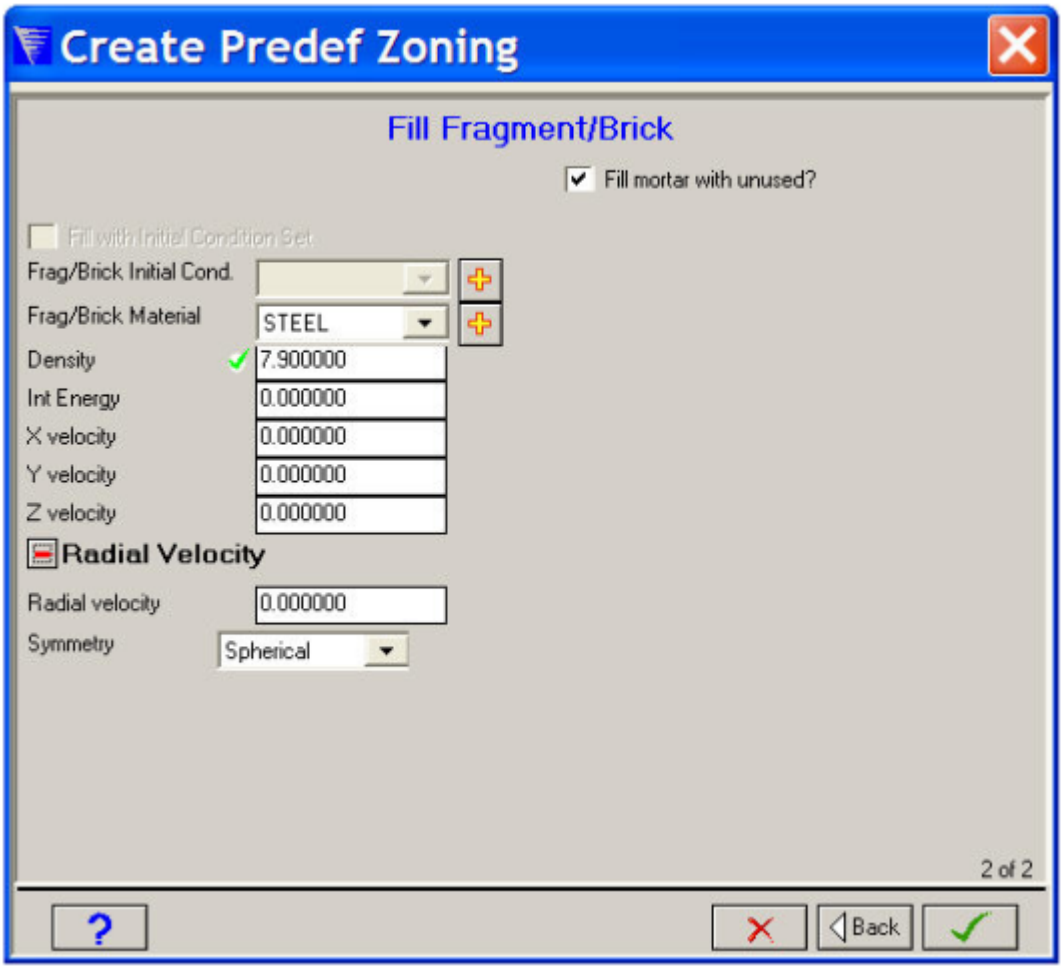
Fragment/brick symmetric about X=0?

Check this box if you want the wall to be symmetric about X=0.

10.9.8. Parts - Zoning - Frag/Bric (Page 2)

Two dialog windows let you build a fragment or brick wall.

This second window lets you fill the wall grid with materials and initial conditions.



Fill mortar with unused?

Check this box if you do not want the mortar to be represented as a material.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

If you check this box, no elements will be generated for the mortar.

If you do not check this box, elements will be generated for the mortar (If the mortar thickness is small compared to the dimension of the brick elements, the time step is likely to be reduced significantly).

Fill with Initial Condition Set

Check this box if you want to fill your wall Part using an Initial Condition Set.

The advantage of filling a Part using an Initial Condition Set is that if you later change one or more parameters in the set, you do not have to refill your Part.

Use the pull-down menu to select the Initial Condition Set you want to use.

You can define new Initial Condition Sets by clicking the **Init.Cond.** button on the Navigation Bar or by clicking next to the option in this window.

Material

If you do not want to use an Initial Condition Set, select the material you want to use to fill your Part from this Pull-down menu.

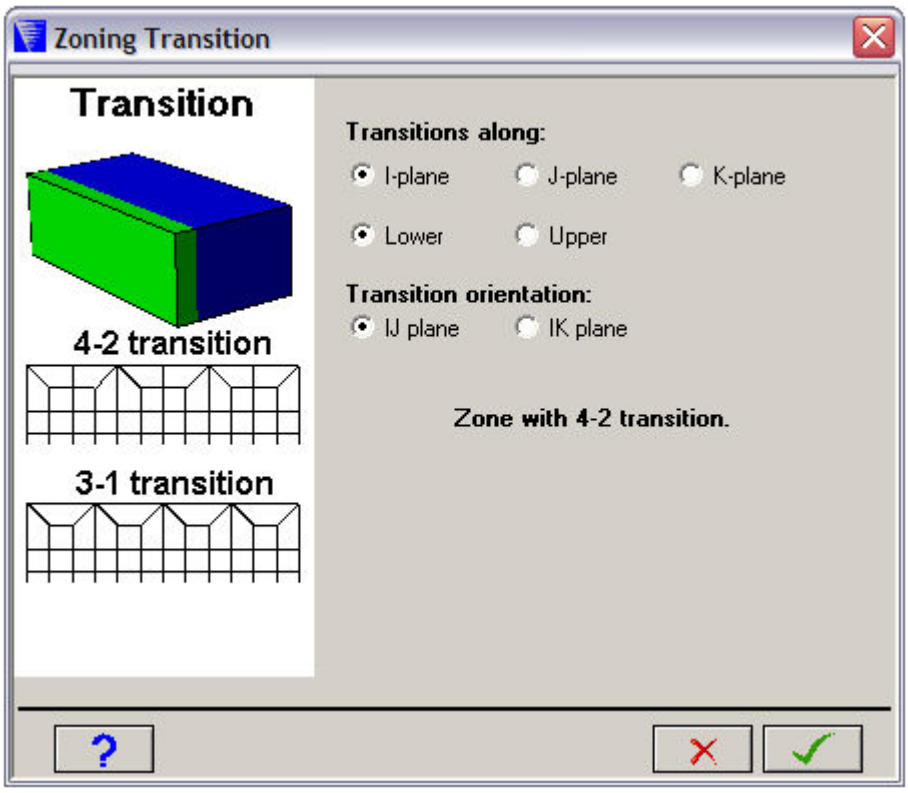
You can define new materials by clicking the **Materials** button on the Navigation Bar or by clicking next to the option in this window.

Parameters

If you do not want to use an Initial Condition Set, enter parameter values for your fill in the fields provided (**Density**, **Energy**, **Velocity**).

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

10.9.9. Parts - Zoning - Transition



This window lets you create a transition at any surface of the block you are zoning.

Transitions along

Specify at which of the six surfaces of the block you want your transition to be created, by selecting and I, J or K-plane and whether it is the corresponding lower or upper surface of the block.

Transition orientation

The orientation of the transition.

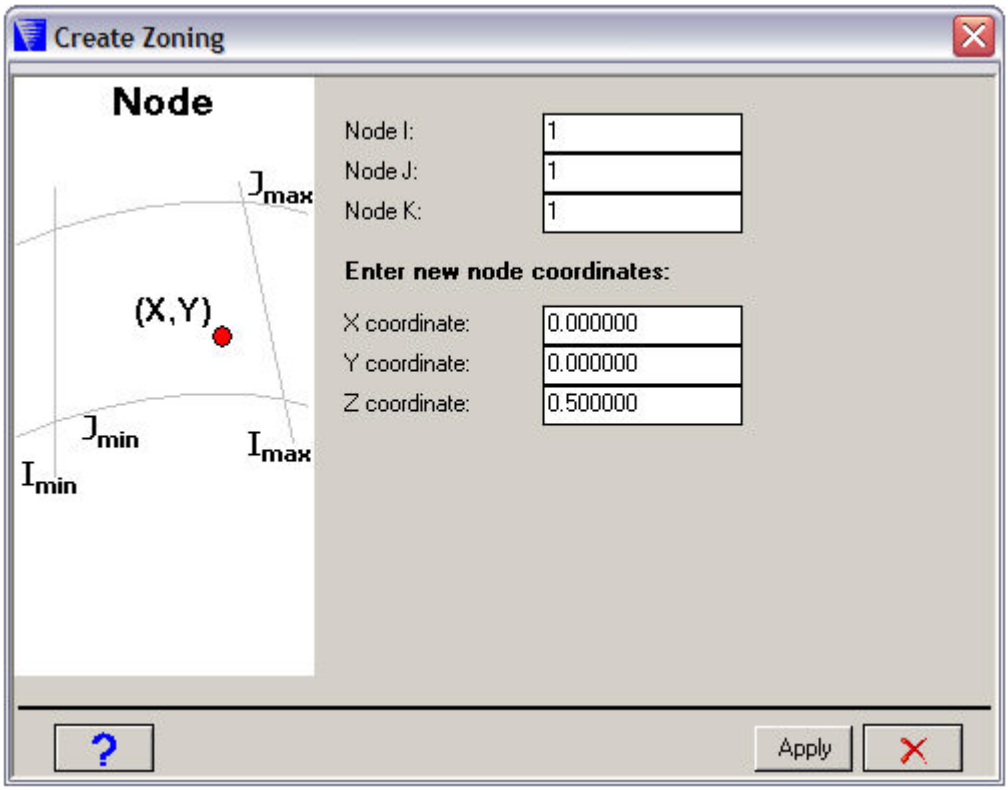
Transition Types

The image above shows you the two transition types you can use, namely a 4-2 transition and a 3-1 transition. To use a particular type, the index range in the direction of the transition orientation must be a multiple of 4 or 3 respectively.

A message will tell if a valid transition type exists for the boundary you have defined. If more than one type is valid, you will be asked to choose the transition type you want to use.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

10.9.10. Parts - Zoning - Node



This window lets you specify the coordinates of a single node.

Node I/J/K

The I, J, and K index to select the node for which you want to specify coordinates.

Old Coordinates

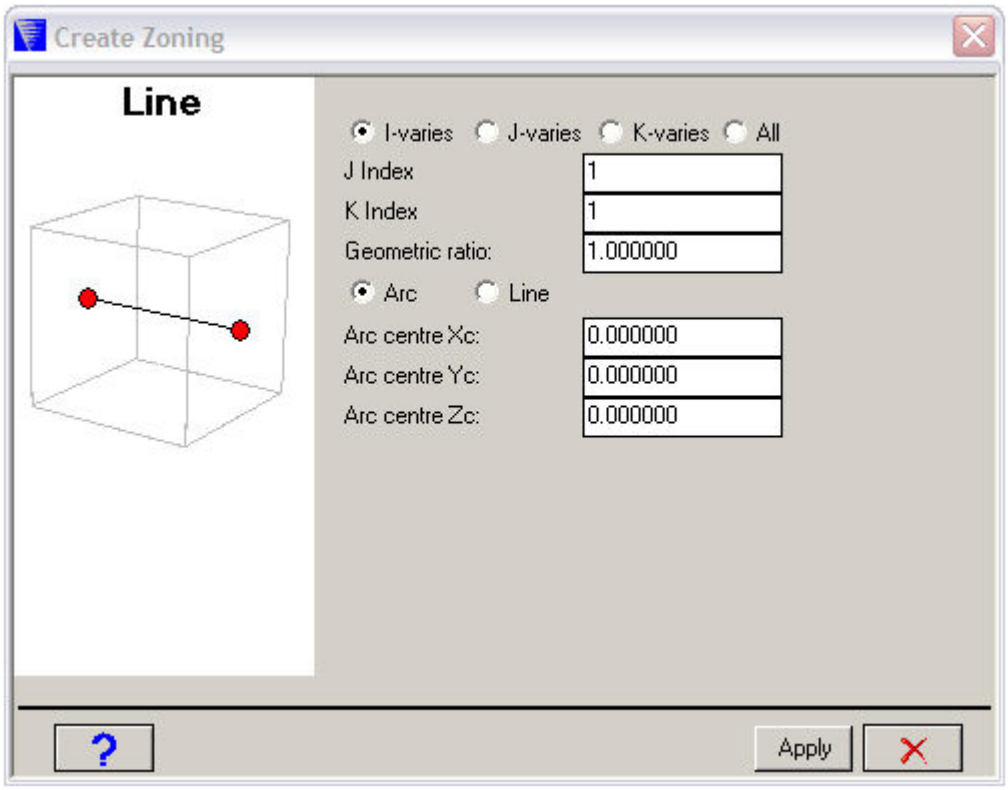
The current (old) coordinates of the selected node are displayed for your information.

New Coordinates

The new coordinates for the selected node.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

10.9.11. Parts - Zoning - Line



This window lets you generate nodes along a line in index space from its two end points. If either of its two end nodes has not been defined, you are asked to input the coordinates of the node.

I/J/K-varies

The index direction of the line you want to generate. The line will be generated across the IJK range in the selected direction.

All

Select this option if you want to generate nodes for all 12 edges of the IJK range you have specified.

Geometric Ratios

Input geometric ratios in the fields provided if you want to grade the zoning instead of generating equally spaced nodes.

Line

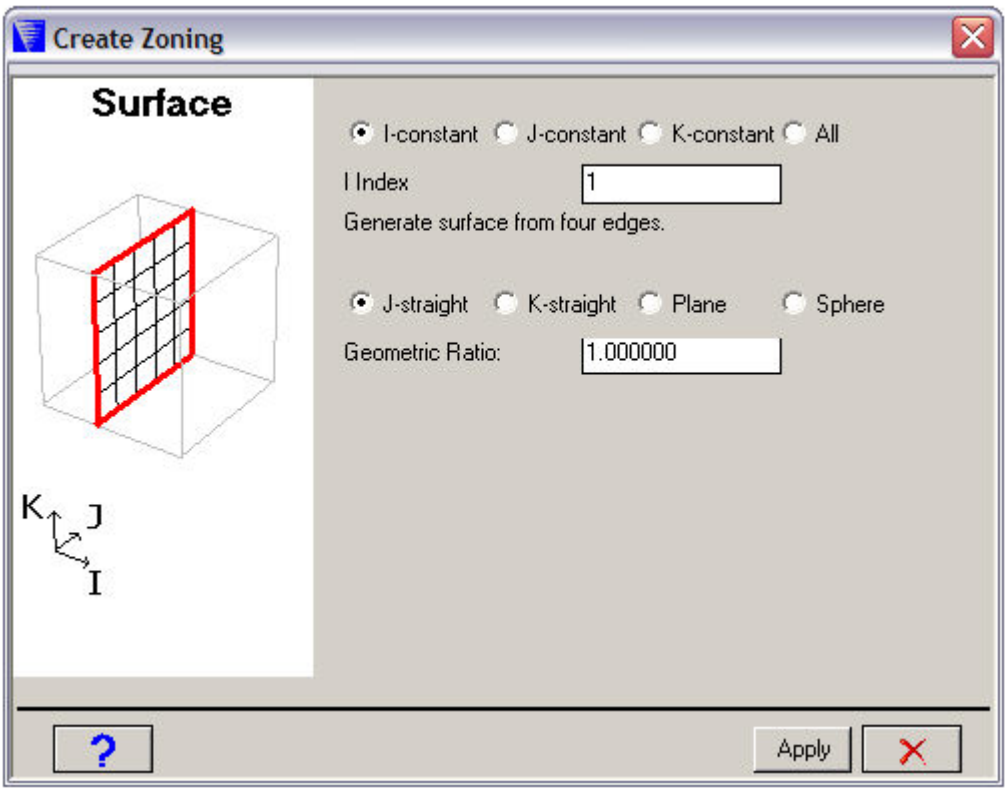
Select this option if you want to generate the nodes along a straight line.

Arc

Select this option if you want to generate the nodes along the arc of a circle (you will be asked to specify the center of the circle).

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

10.9.12. Parts - Zoning - Surface



This window lets you zone a surface of the IJK region you selected (i.e. an index plane with I, J, or K constant) from nodes already defined along its four edges.

I/J/K-constant

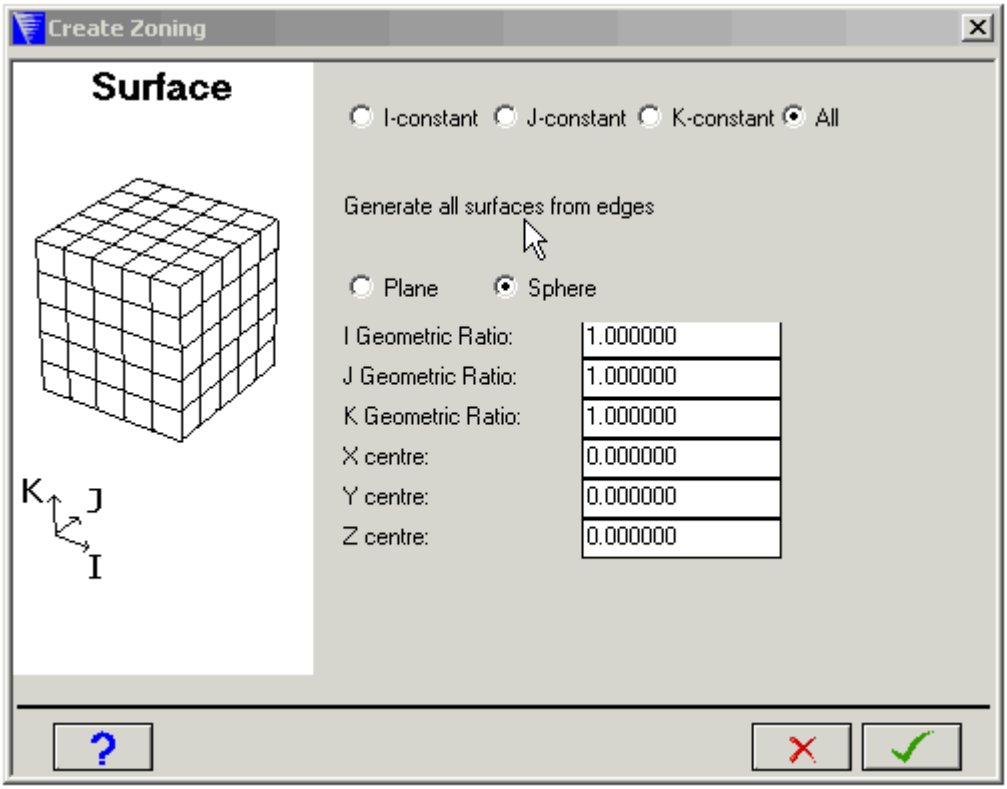
Select whether you want to zone a surface with a constant I, J, or K index.

All

Select this option if you want to zone **all six outer surfaces** of the IJK range you have selected.

The dialog window will then look like this:

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Index (not for All)

The I, J, or K index for the surface.

I/J/K-straight (not for All)

Select one of these options if you want surface nodes to be generated along straight I, J, or K lines.

Plane

Select this option if you want surface nodes to be generated equipotentially in a plane.

Sphere

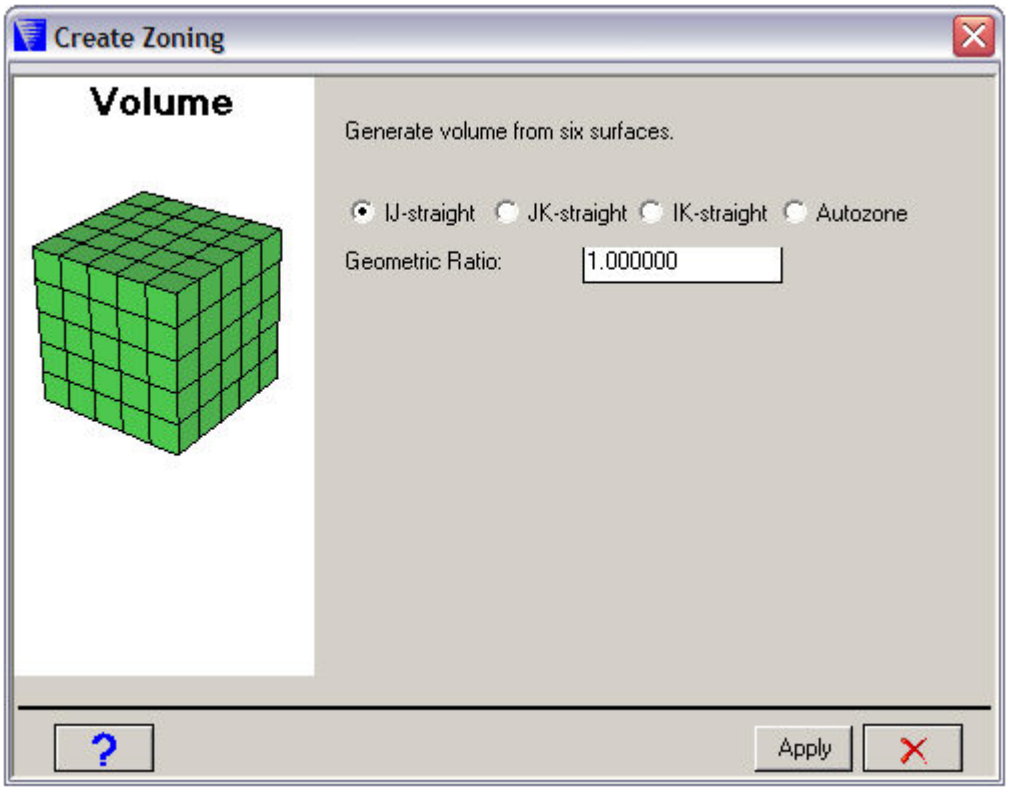
Select this option if you want surface nodes to be generated equipotentially on the surface of a sphere (you will be asked to specify the center of the sphere).

Geometric Ratios

Set geometric ratios in the fields provided if you want to grade the zoning instead of equally spacing the nodes.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

10.9.13. Parts - Zoning - Volume



This window lets you zone the interior volume of the IJK region you selected from nodes already defined on its six faces.

IJ-straight

Select this option if you want to generate the interior volume so that nodes with the same (I,J) indices lie equally spaced along a straight line

JK-straight

Select this option if you want to generate the interior volume so that nodes with the same (J,K) indices lie equally spaced along a straight line

IK-straight

Select this option if you want to generate the interior volume so that nodes with the same (I,K) indices lie equally spaced along a straight line

Autozone

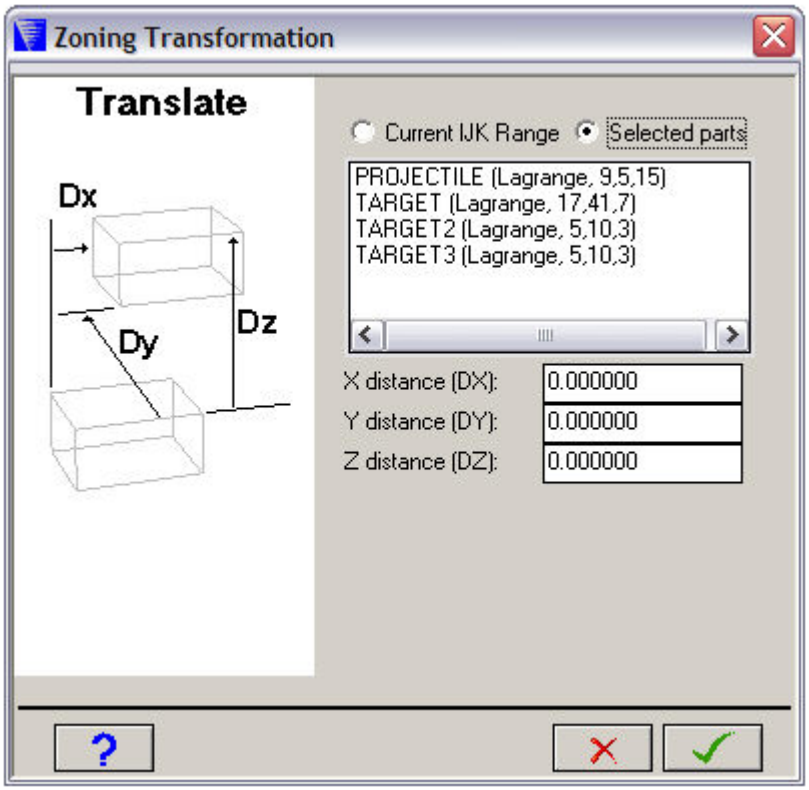
Select this option if you want to generate the interior volume so that all nodes are spaced equi-potentially.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Geometric Ratio

Enter a geometric ratio in this field if you want to grade the zoning instead of equally spacing the nodes.

10.9.14. Parts - Zoning - Translate



This window lets you translate nodes in XYZ space.

Current IJK Range

Select this option to apply the translation to all nodes in the current IJK Range.

Selected Parts

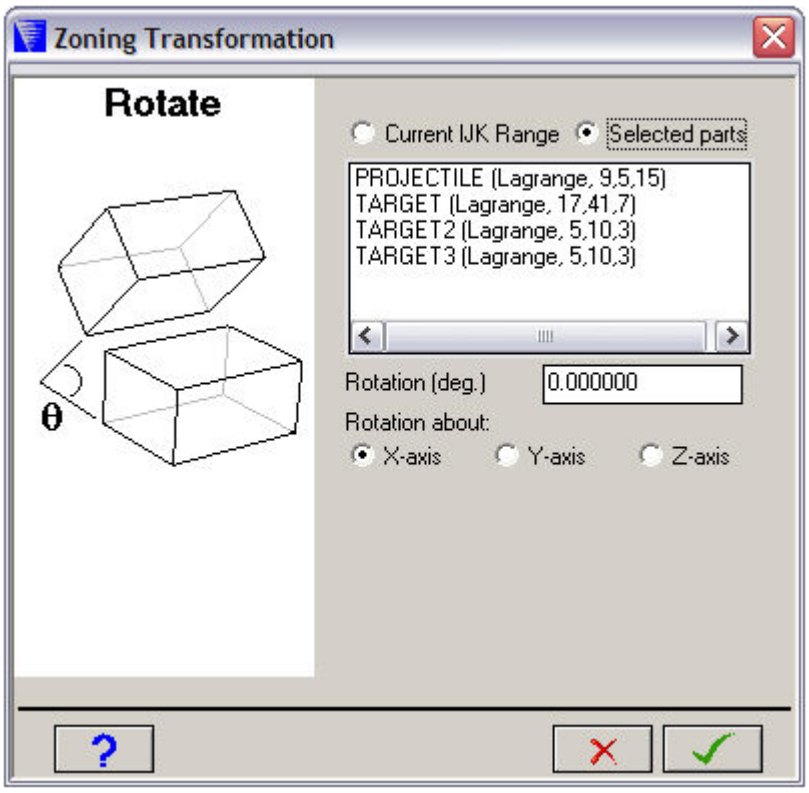
Select this option to apply the translation to all nodes of selected Parts. A box lets you select one or more Parts.

X/Y/Z distance

The X, Y, and Z components of the translation.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

10.9.15. Parts - Zoning - Rotate



This window lets you rotate nodes about an axis in XYZ space.

Current IJK Range

Select this option to apply the rotation to all nodes in the current IJK Range.

Selected Parts

Select this option to apply the rotation to all nodes of selected Parts. A box lets you select one or more Parts.

Rotation

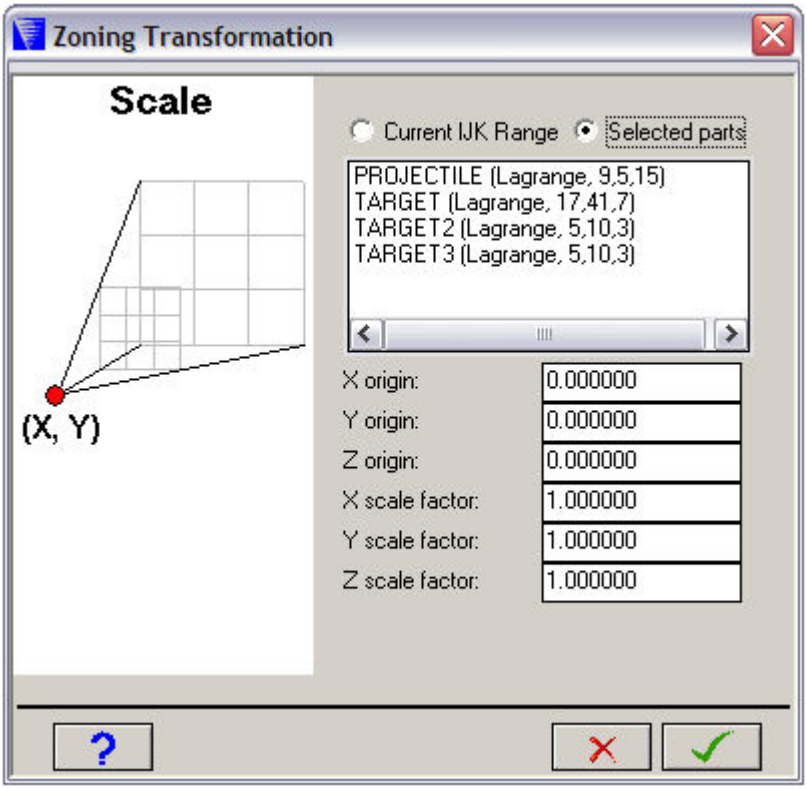
The rotation angle in degrees.

X/Y/Z-axis

The axis about which the nodes will be rotated.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

10.9.16. Parts - Zoning - Scale



This window lets you scale nodes relative to an origin in XYZ space.

Current IJK Range

Select this option to apply the rotation to all nodes in the current IJK Range.

Selected Parts

Select this option to apply the rotation to all nodes of selected Parts. A box lets you select one or more Parts.

Origin

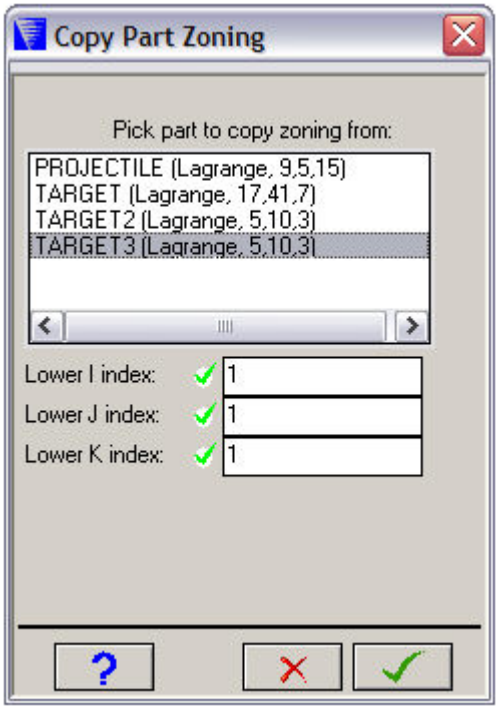
The X, Y, and Z coordinates of the origin for the scaling.

Scale Factors

The X, Y, and Z scale factors.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

10.9.17. Parts - Zoning - Copy



This window lets you copy zoning from an existing Part to the current IJK Range

Pick part to copy

The Part from which you want copy zoning.

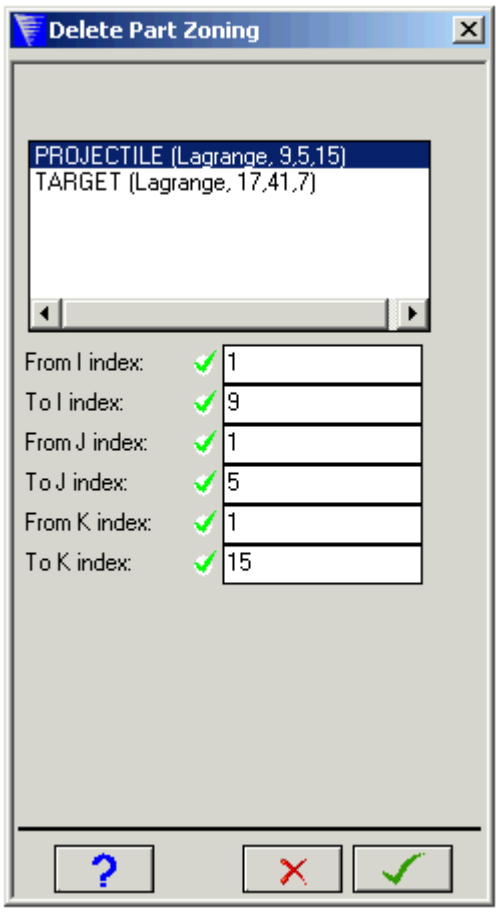
Lower I/J/K index

The lower I, J, and K indices of the block from which you want to copy.

A block starting at these lower indices will be copied to the current IJK Range.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

10.9.18. Parts - Zoning - Delete



This window lets you delete the zoning in a Part.

Parts List

The Part in which you want to delete some zoning.

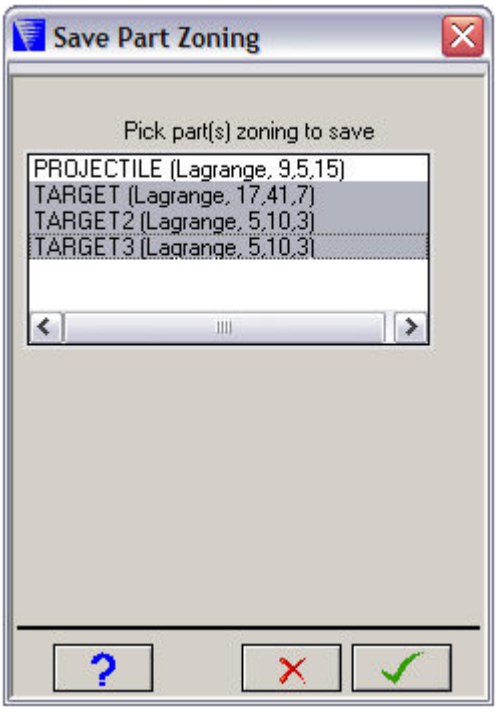
Index Range

The index range of the nodes whose zoning you want to delete.

The coordinates of all nodes in the index range will be set to **Undefined** (the default state).

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

10.9.19. Parts - Zoning - Save



This window lets you temporarily save the zoning for one or more Parts.

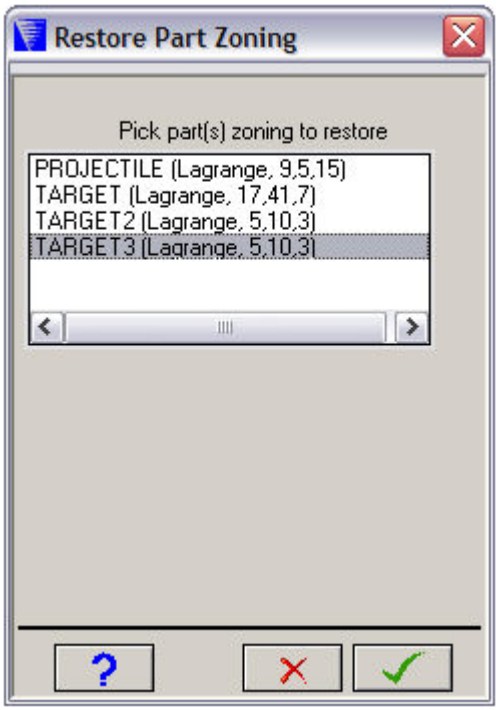
This is a useful option if you are about to do some detailed zoning changes and want to ensure you can return to your original zoning if you make a mistake.

Clicking the Restore button in the Parts-Zoning panel lets you restore the saved zoning.

Select the Parts whose zoning you want to save and click .

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

10.9.20. Parts - Zoning - Restore



This window lets you restore zoning you have temporarily saved for one or more Parts. (Clicking the Save button in the Parts-Zoning panel lets you temporarily save zoning for one or more Parts.)

Select the Parts whose zoning you want to restore and click .

10.9.21. Parts - Zoning - Refine

This window lets you refine structured Lagrange, ALE, Euler-Godunov and Euler-FCT parts. However, the part must satisfy the following requirements in order to utilize this new refining facility:

- The part must be a 'box' type object.
- The X, Y, and Z coordinates of the grid nodes must increase with the I, J, and K indices respectively.

I/J/K Factor

Enter the Refine Factor for the I/J/K index; that is, number of the new cells created from an original single cell in that direction.

The refining algorithms are by default applied to all cells within a part. In order to limit the range of cells within a part that undergo refinement, you can select an appropriate IJK-RANGE first. In addition it should be noted that the following limitations apply to the refinement of Euler and ALE parts.

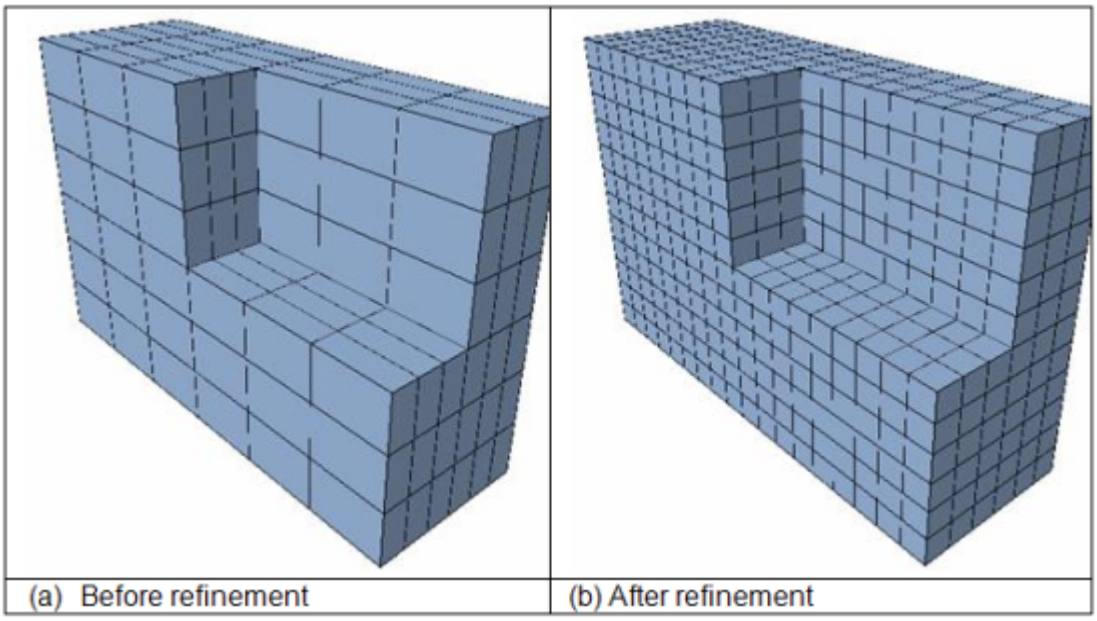
- For simplicity with multi-material Euler-Godunov grids, the material volume fraction of an original cell is assigned to all the new cells created from that original cell. Thus the material interface on the new grid may not be exactly the same as that on the original grid.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

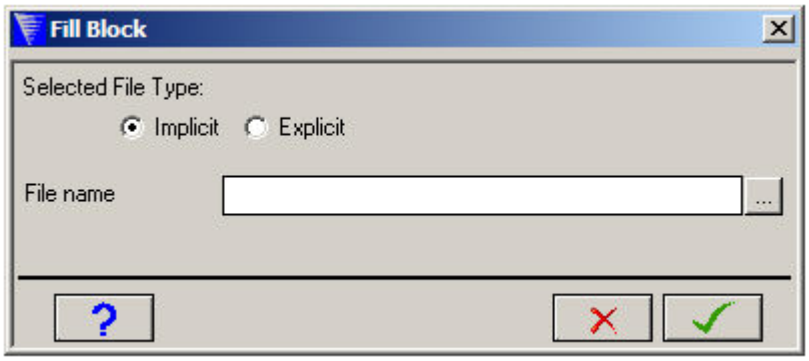
- For ALE grids, the original motion constraint of the ALE nodes is not retained during the refining procedure. The motion constraint for all the new nodes is set to Lagrange, even if the constraint of the original node is, for example, Equipotential. You will therefore need to redefine the motion constraints for ALE parts following the refining process.

The cell index for time history gauge points is also updated during the refinement. Furthermore, the mass is always conserved between the original and refined parts. However, the momentum and kinetic energy in Lagrange and ALE parts may not be conserved exactly because the mesh refinement modifies the nodal masses and thus affects the momentum and kinetic energy calculation. For Euler-Godunov and Euler-FCT parts, since the cell mass is used to calculate both the kinetic energy and momentums, refinement does not affect the calculation. The kinetic energy and momentums should therefore always be conserved.

An example of the refinement of a Lagrangian part is shown below. Note that the refinement factors IFACT, JFACT and KFACT have been set to 1, 2 and 3 respectively.



10.9.22. Parts - Zoning - Fill Block



This window lets you import complicated zoning and fill information, such as a city center consisting of many buildings of different size. The option is only available for Euler-FCT parts.

After browsing for an input file (extension `.blk`), select the format of the external data.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- **Implicit** format – Autodyn reads the fill flag of Euler-FCT cells in a certain predefined sequence. The fill flags of ALL the cells in the input region must therefore be defined in the data file.
- **Explicit** format – Autodyn reads the cell index together with its fill flag. Hence, only the buildings (unused) cells and the cells with target points need to be included in the data file.

The external file imported into Autodyn using the Fill Block option should have the extension `.blk`, for example `block0.blk`. Note that this file name is not necessarily the same as the ident of the Autodyn model that is being pre-processed.

The Fill Block command also permits extra air space to be created beyond the input region. For example, to allow a pressure wave to propagate over the tallest building in the model and interact with the buildings behind, Fill Block can be used to create additional air space above the building.

10.9.22.1. Parts - Zoning - Fill Block Implicit

IMPLICIT	Fill type
IMAX, JMAX, KMAX	Maximum I, J, K indices of input region
XMIN, YMIN, ZMIN	Minimum co-ordinates of input region (corresponding to the node at I=J=K=1)
DX, DY, DZ	X, Y, Z zone dimensions for input region
XRAT, YRAT, ZRAT	Ratios used to extend grid beyond the original input region. For example, if XMAX is the maximum X-coordinate of the input region and XRAT=0.5, the generated grid would be extended to XMAX + 0.5 (XMAX-XMIN) in the X-direction, with geometric zoning being used automatically for cells above XMAX. The maximum I index of the part will be larger than the IMAX defined in the data file, hence extra elements are created to include the extended air space.
SIE	Unit-dependent specific internal energy for the air (2.068x105 is usually OK for the mm/mg/ms units)
IFILL	Fill flag of each cell in input region (K varying faster than J, J varying faster than I. Input 20 cells per line of dataIFILL= 0 Fill cell with unused IFILL= 1 Fill cell with air IFILL= 2 Fill cell with air and target point

An example of a `.blk` file using the implicit format is shown below. Note that the IFILL flag has to be defined for all elements in the part. The resulting filled Euler-FCT grid is shown in Figure 2.2.1.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

```
IMPLICIT
6  6  6
0.0000E+00          0.0000E+00          0.0000E+00
0.1000E+03          0.1000E+03          0.1000E+03
0.4400E+00          0.4400E+00          0.4400E+00
2.0680E+05
1  1  1  1  1  1  1  1  1  1  1  1  1  1  1  1  1  1  1  1
1  1  1  1  1  1  1  1  1  1  1  1  1  1  1  1  1  1  1  2
0  0  0  0  0  0  0  0  0  0  0  0  0  0  0  0  0  0  0  0
2  1  1  1  1  1  1  1  1  1  1  1  1  1  1  1  1  1  1  1
1  1  1  1  1  1  1  1  1  1  1  1  1  1  1  1  1  1  1  1
1  1  1  1  1  1  1  1  1  1  1  1  1  1  1  1  1  1  1  1
1  1  1  1  1
```

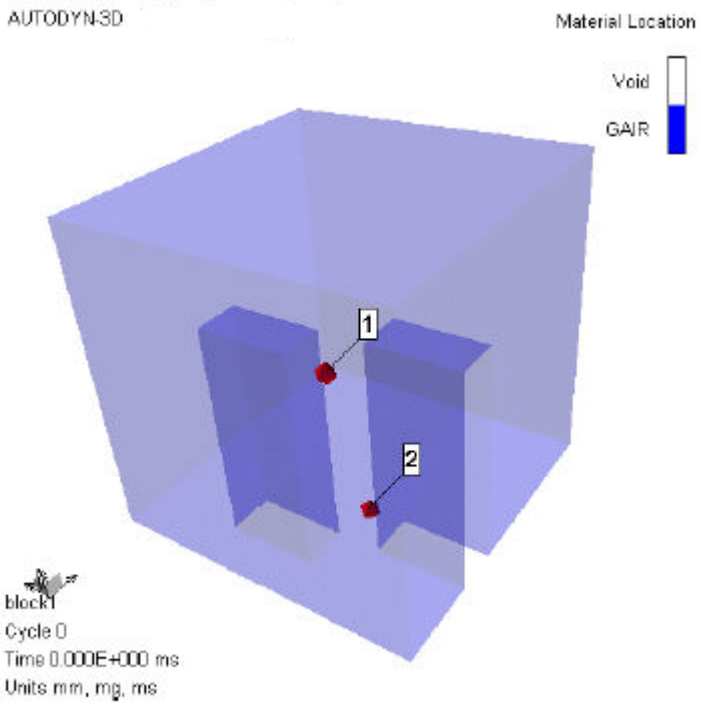


Figure 2.2.1 Example Euler-FCT grid using the Implicit Format

10.9.22.2. Parts - Zoning - Fill Block Explicit

EXPLICIT	Fill type
IMAX, JMAX, KMAX	Maximum I, J, K indices of input region
XMIN, YMIN, ZMIN	Minimum co-ordinates of input region (corresponding to the node at I=J=K=1)
DX, DY, DZ	X, Y, Z zone dimensions for input region
XRAT, YRAT, ZRAT	Ratios used to extend grid beyond the original input region. For example, if XMAX is the maximum X-coordinate of the input region and XRAT=0.5, the generated grid would be extended to XMAX + 0.5 (XMAX-XMIN) in the X-direction, with geometric zoning being used automatically for cells above XMAX. The maximum I index of the part will be larger than the

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

	IMAX defined in the data file, hence extra elements are created to include the extended air space
SIE	Unit-dependent specific internal energy for the air (2.068x105 is usually OK for the mm/mg/ms units)
I, J, K, IFILL	I, J, and K index of the cell and its fill flag. One cell per line of input. IFILL = 0 Fill cell with unused IFILL = 1 Fill cell with air (default) IFILL = 2 Fill cell with air and target point

An example of a `.blk` file using the explicit format is shown below. Note that only elements with an IFILL flag of 0 or 2 need to be defined. The default of 1 is assumed for all other cells. The resulting filled Euler-FCT grid is shown in Figure 2.2.2.

```
EXPLICIT
11  11  11
0.0000E+00          0.0000E+00          0.0000E+00
0.1000E+03          0.1000E+03          0.1000E+03
0.5000E+00          0.5000E+00          0.5000E+00
2.0680E+05
6   6    2   0
6   6    3   0
6   6    4   0
6   6    5   0
6   6    6   0
6   6    7   0
6   6    8   0
6   6    9   0
6   6    10  0
6   6    11  0
5   6    2   2
5   6    3   2
5   6    4   2
5   6    6   2
5   6    7   2
5   6    8   2
5   6    9   2
5   6    10  2
5   6    11  2
END
```

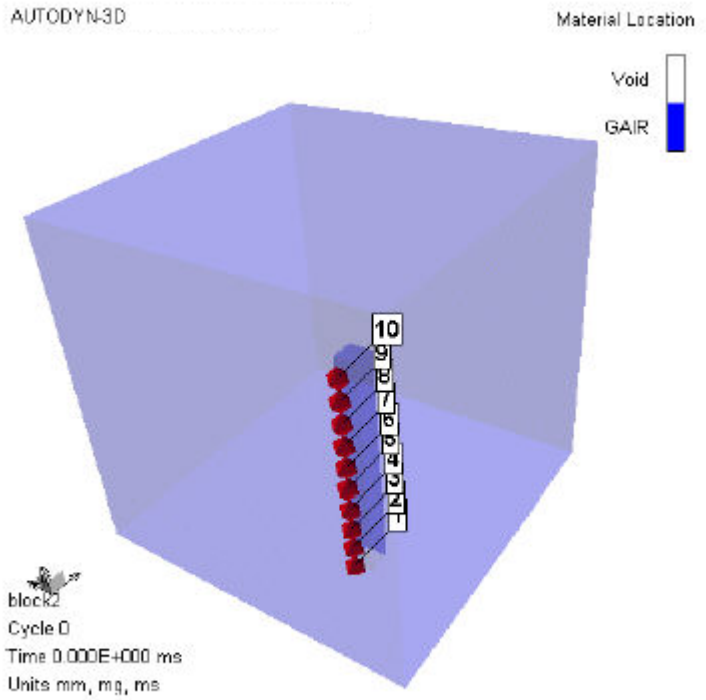
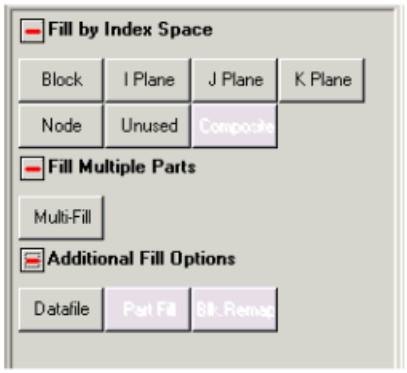


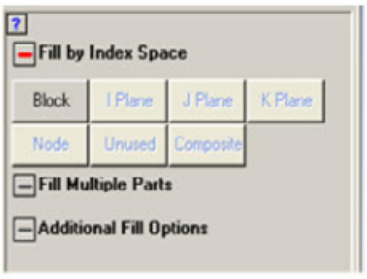
Figure 2.2.2 Example Euler-FCT grid using the Explicit Format

10.10. Parts - Fill

Clicking the **Fill** button in the **Parts panel** displays the following dialog in the lower region of the panel :



Structured



Unstructured

This dialog lets you fill the selected Part with materials and initial conditions.

The only option available for unstructured parts is to fill the whole part. When an unstructured parts only needs to be filled partially, the Group option can be used to create a selected group of elements which then can be filled with materials or initial conditions.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Fill by Index Space

Block

Fill a block in index space.

I/J/K Plane

Fill a plane in index space.

Node

Fill a node.

Unused

Fill a block in index space with unused elements.

Composite

Fill a composite shell.

Fill Multiple Parts

Multi-Fill

Fill multiple Parts.

Additional Fill Options

Datafile

Read or write a datafile for General Remapping.

Part Fill

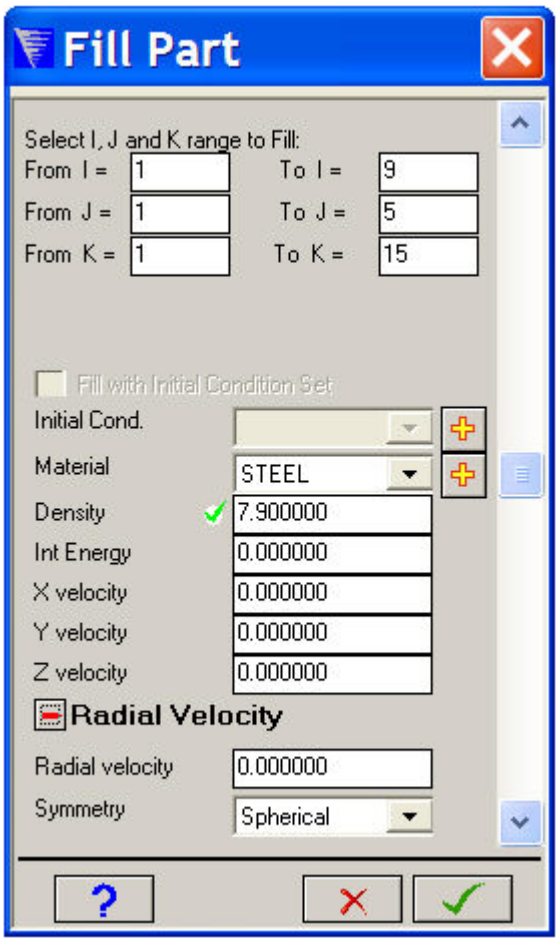
Fill an Euler Part using Lagrange or Fill Parts.

Block Remap

Read or write a datafile for Fast Block Remapping.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

10.10.1. Parts - Fill - Block



This window lets you fill an index block within the current Part.

Select I, J, and K range to Fill

The I, J, and K range that you want to fill.

Fill with Initial Condition Set

Check this box if you want to fill the block using an Initial Condition Set.

The advantage of filling a Part using an Initial Condition Set is that if you later change one or more parameters in the set, you do not have to refill your Part.

Use the pull-down menu to select the Initial Condition Set you want to use.

You can define new Initial Condition Sets by clicking the **Init.Cond.** button on the Navigation Bar or by clicking next to the option in this window.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Material

If you do not want to use an Initial Condition Set, select the material you want to use to fill your Part from this Pull-down menu.

You can define new materials by clicking the **Materials** button on the Navigation Bar or by clicking next to the option in this window.

Parameters

If you do not want to use an Initial Condition Set, enter parameter values for your fill (**Density**, **Energy**, **Velocity**).

Radial Velocity

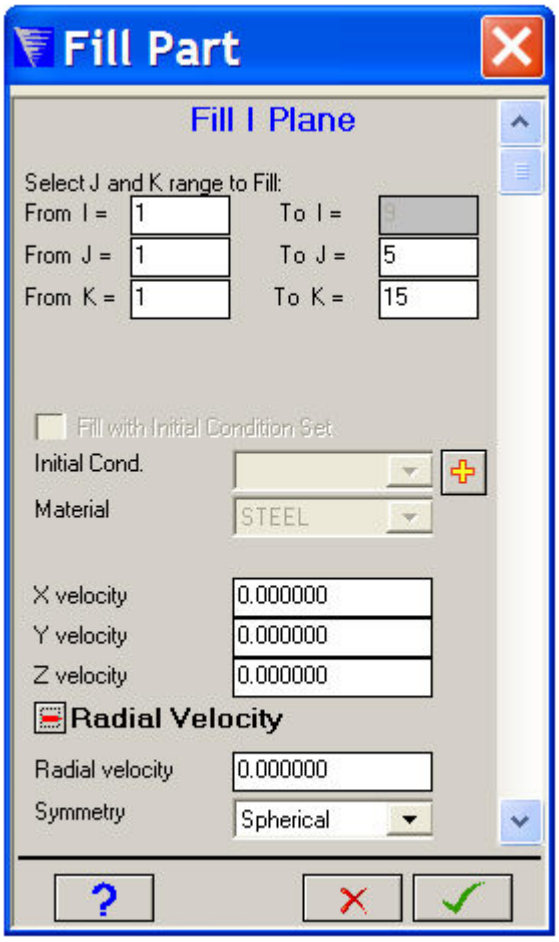
The radial velocity and choose between spherical or cylindrical symmetry. For cylindrical symmetry the direction of the cylinder axis must also be specified.

Axial Velocity

The axial velocity and select the origin and direction of the axis.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

10.10.2. Parts - Fill - I / J / K Plane



This window lets you fill an I, J, or K plane of the current Part with initial conditions.

The example shown is for filling an I plane (input for the other planes is similar).

Select range to Fill

The I index for the plane you want to fill and the J and K range to fill in this plane.

Fill with Initial Condition Set

Check this box if you want to fill the plane using an Initial Condition Set.

The advantage of filling a Part using an Initial Condition Set is that if you later change one or more parameters in the set, you do not have to refill your Part.

Use the pull-down menu to select the Initial Condition Set you want to use.

You can define new Initial Condition Sets by clicking the **Init.Cond.** button on the Navigation Bar or by clicking next to the option in this window.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

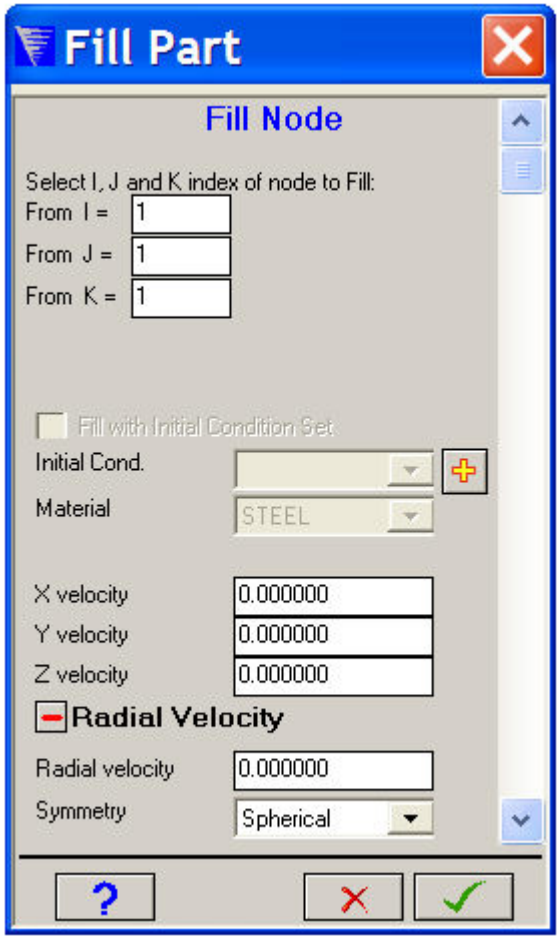
Material

Because you are only filling a plane, no material fill is done, only initial conditions for nodes are set (that is, velocities)

Parameters

If you do not want to use an Initial Condition Set, enter parameter values for your fill (Velocity only).

10.10.3. Parts - Fill - Node



This window lets you fill an I, J, or K plane of the current Part with initial conditions.

The example shows is for filling an I plane (input for the other planes is similar).

Select the node to Fill

The I, J, and K indices for the node you want to fill.

Fill with Initial Condition Set

Check this box if you want to fill the node using an Initial Condition Set.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

The advantage of filling a Part using an Initial Condition Set is that if you later change one or more parameters in the set, you do not have to refill your Part.

Use the pull-down menu to select the Initial Condition Set you want to use.

You can define new Initial Condition Sets by clicking the **Init.Cond.** button on the Navigation Bar or by clicking next to the option in this window.

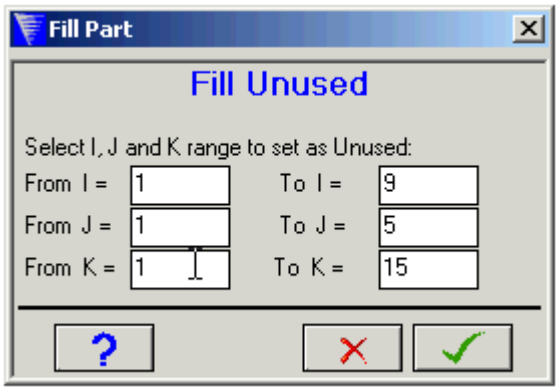
Material

Because you are only filling a node, no material fill is done, only initial conditions for the node are set (that is, velocities)

Parameters

If you do not want to use an Initial Condition Set, enter parameter values for your fill (Velocity only).

10.10.4. Parts - Fill - Unused

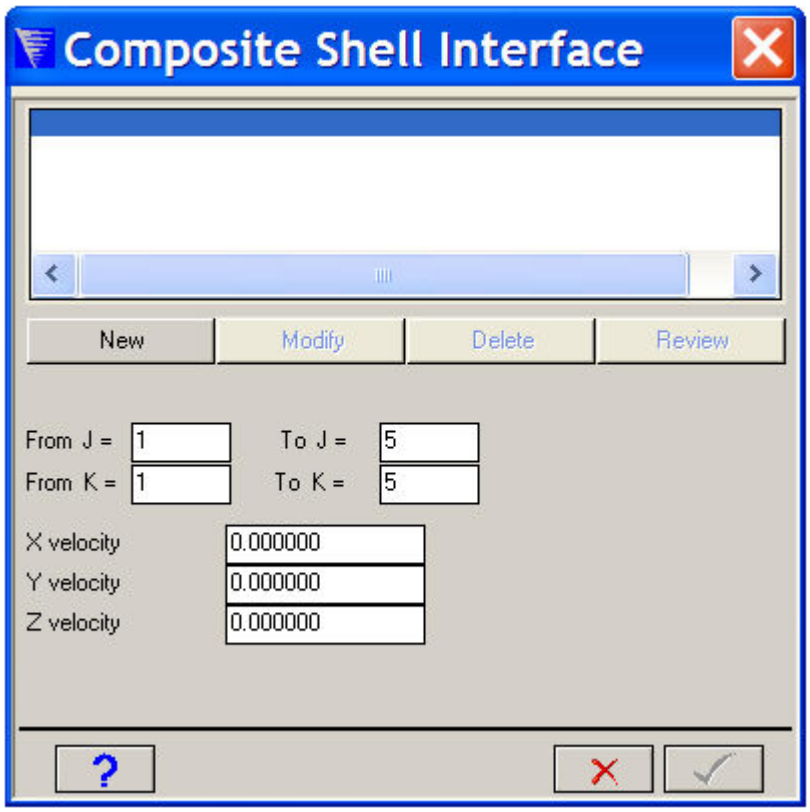


This window lets you fill a block in index space with **Unused** elements (previous materials occupying the elements are removed).

Enter the index range of the block you want to fill and click .

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

10.10.5. Parts - Fill - Composite



This window lets you define a shell as a composite shell, consisting of a number of different layers. The composite allows efficient modeling of thin composite materials which are subjected to structural, rather than shock type loading.

New

Create a new shell layer definition.

Modify

Modify an existing shell layer definition.

Delete

Delete an existing shell layer definition.

Review

Review the existing shell layer definition.

Range

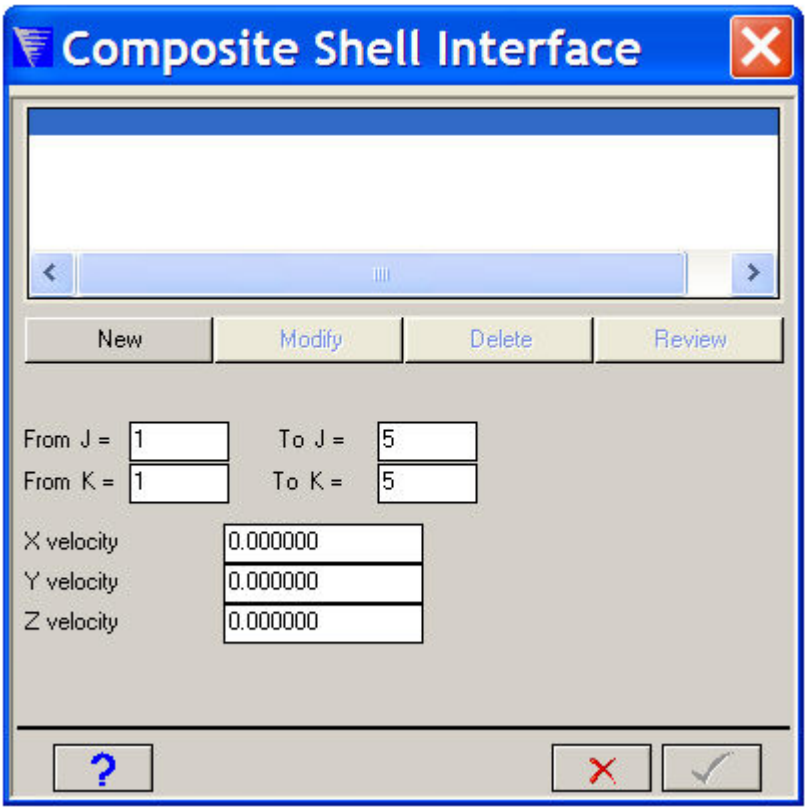
The J and K range of the shell to apply the selected layer definition.

Velocity

The initial velocity of the shell.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

10.10.5.1. Parts - Fill - Composite - New



This window lets you create an existing layer definition.

Number of Layers

Select the number of layers for the composite layer definition.

Layer definition name

Enter a name for the layer definition.

Material

Select the material for each material layer.

Direction

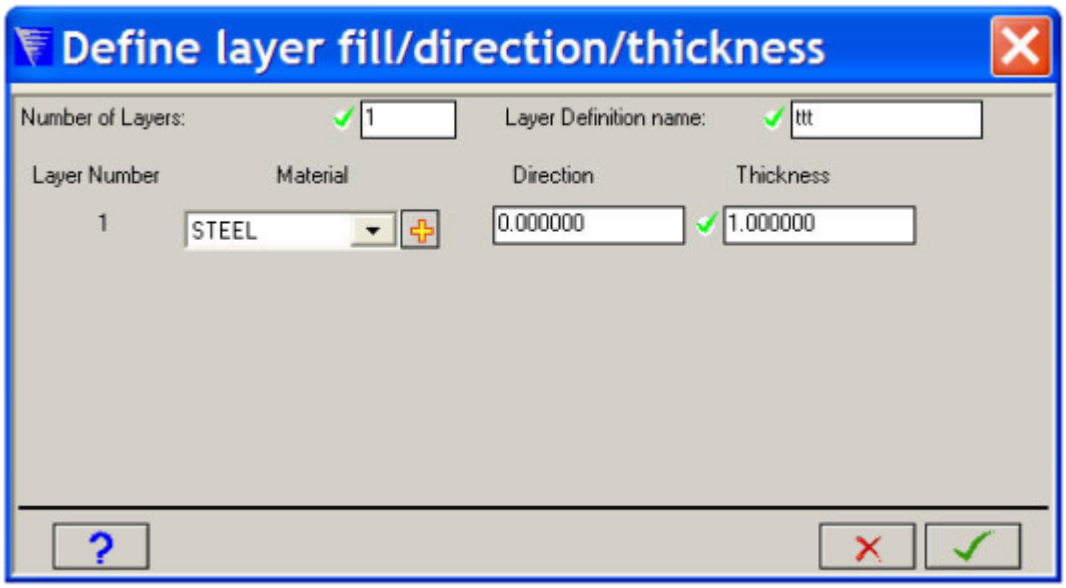
For orthotropic materials, enter a material direction in degrees.

Thickness

The thickness for each material layer.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

10.10.5.2. Parts - Fill - Composite - Modify



This window lets you modify an existing layer definition.

Number of Layers

The number of layers for the composite layer definition.

Layer definition name

The name for the layer definition.

Material

The material for each material layer.

Direction

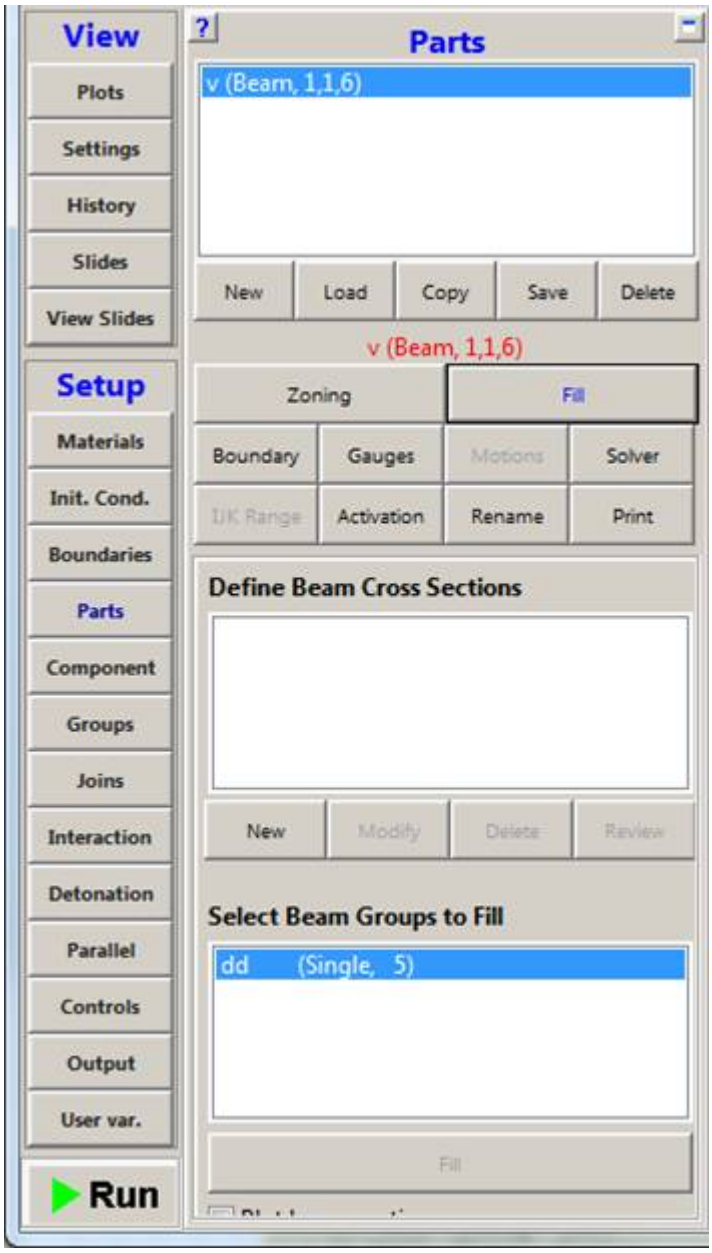
For orthotropic materials, enter a material direction in degrees.

Thickness

The thickness for each material layer.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

10.10.6. Parts – Fill – Beam



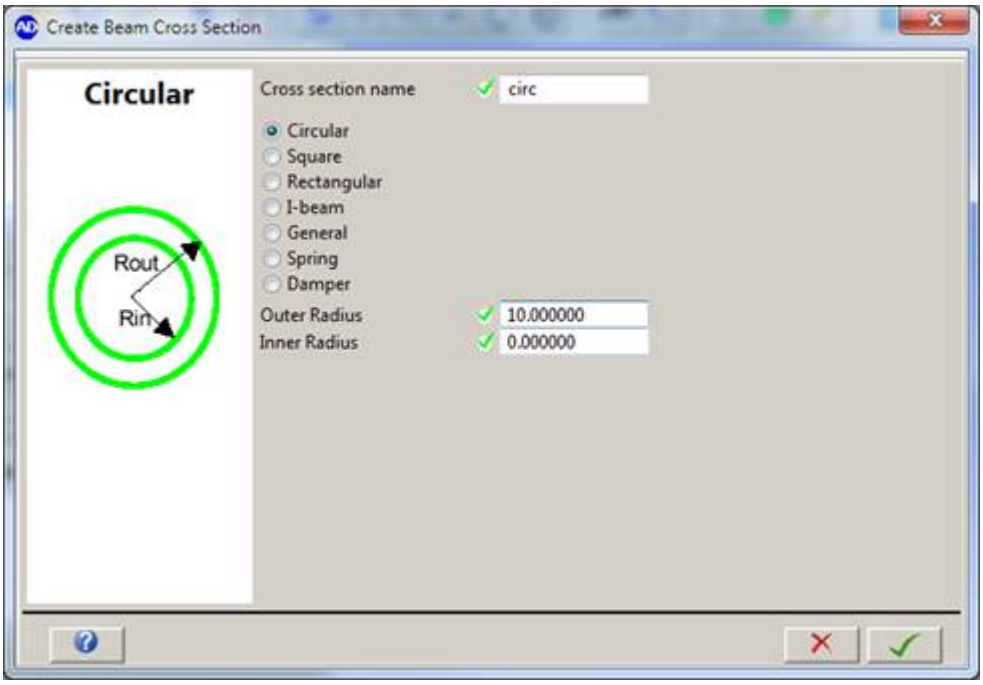
This window lets you create cross section definitions to fill beam groups using the New option, or modify existing cross section definitions using the Modify option.

The available cross sections are Circular, Square, Rectangular, I-beam and General beam, as well as Spring and Damper cross section definitions to model springs and dampers using beam parts.

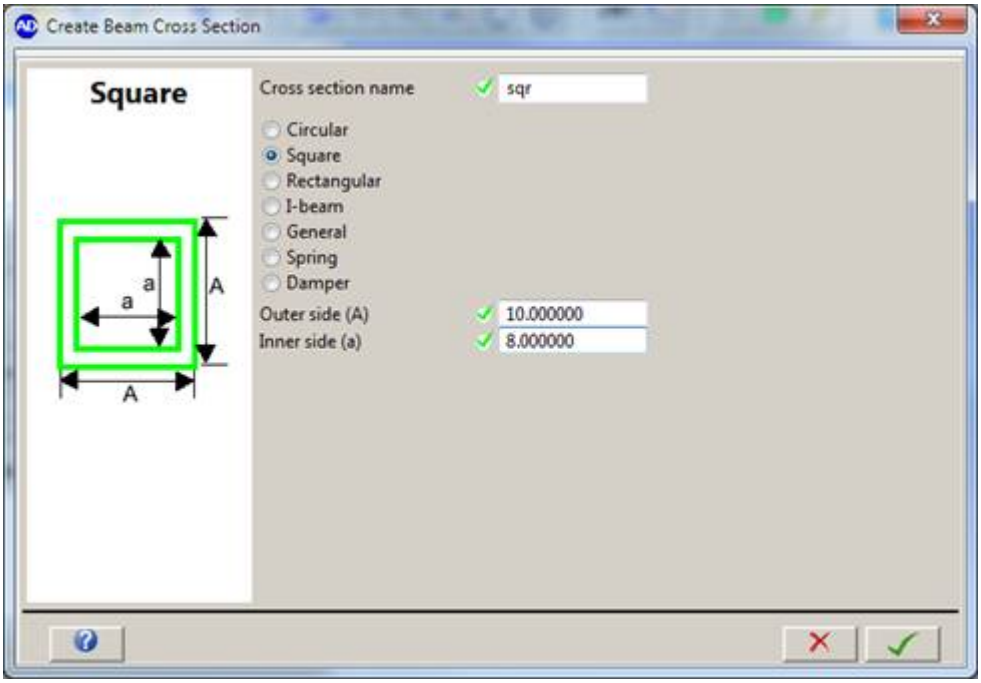
10.10.6.1. Available Beam Cross Sections

Circular – Inner and outer radius of the circular cross section needs to be defined.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

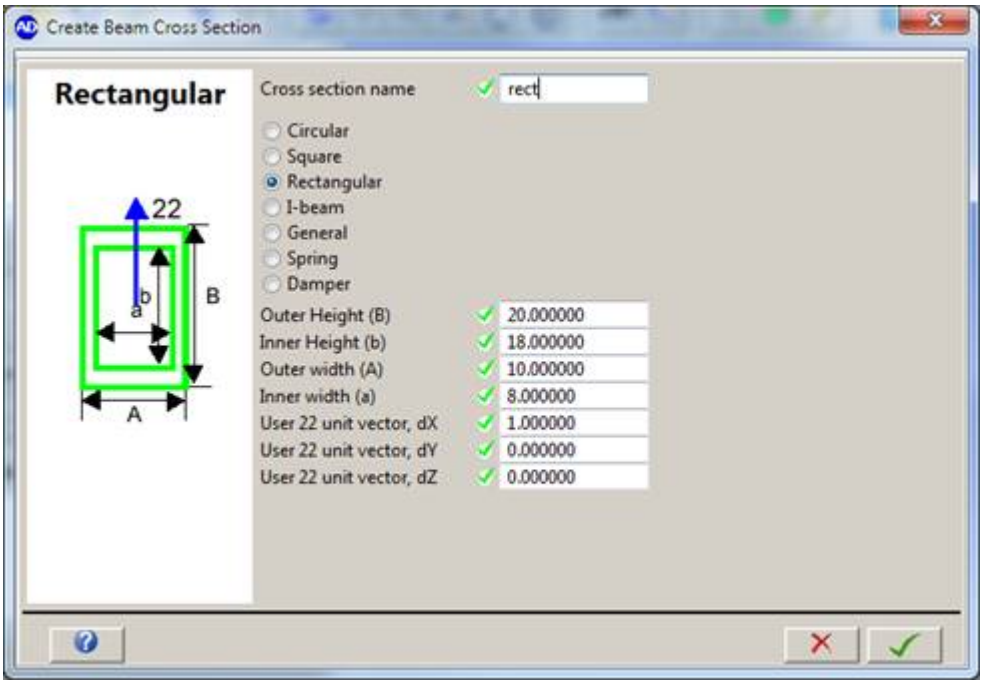


Square – Outer and inner dimension of the square cross section needs to be defined.

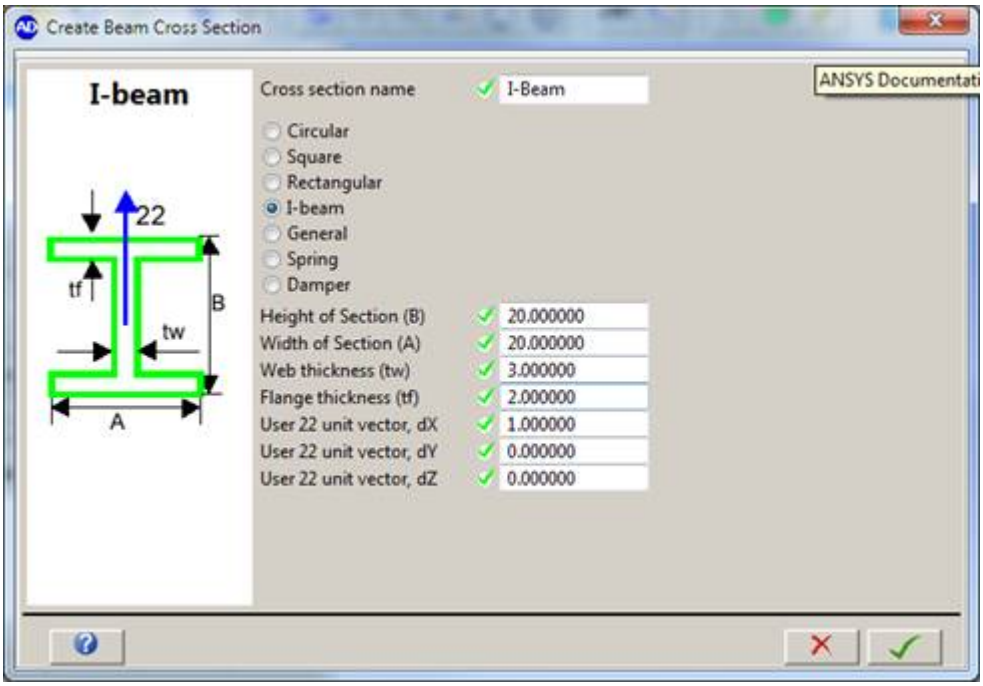


Rectangular – Outer and inner height and width of the rectangular cross section needs to be defined, as well as the unit vector 22.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

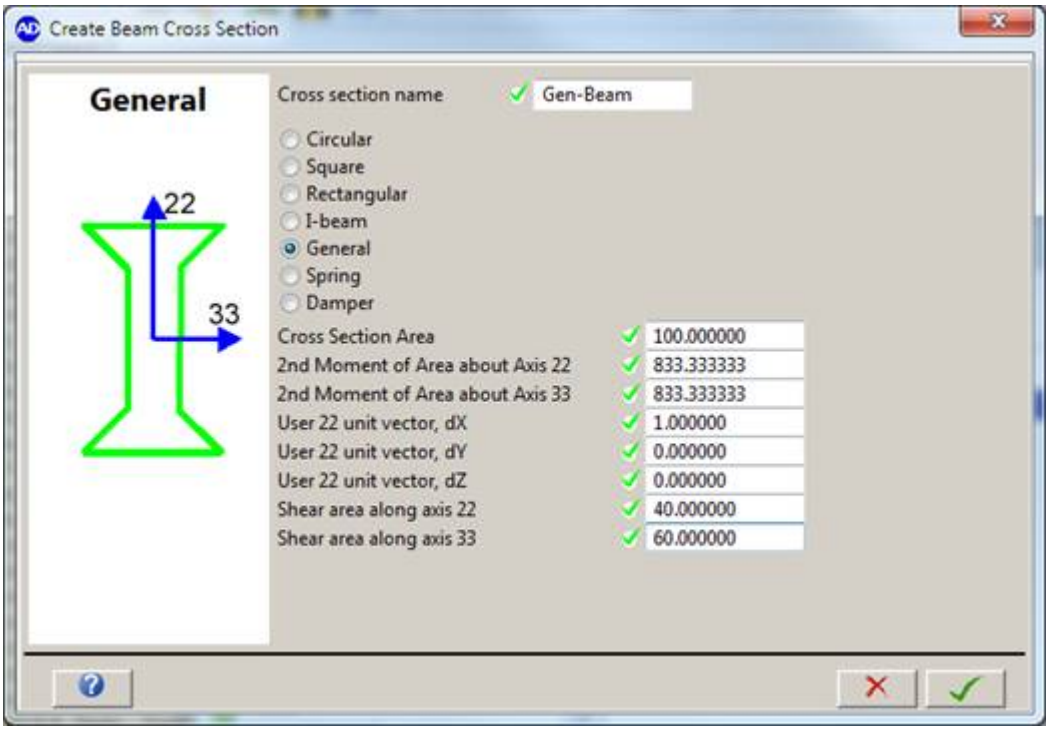


I-beam – Height and width, and thickness of web and flange of the I-beam needs to be defined, as well as the unit vector 22.



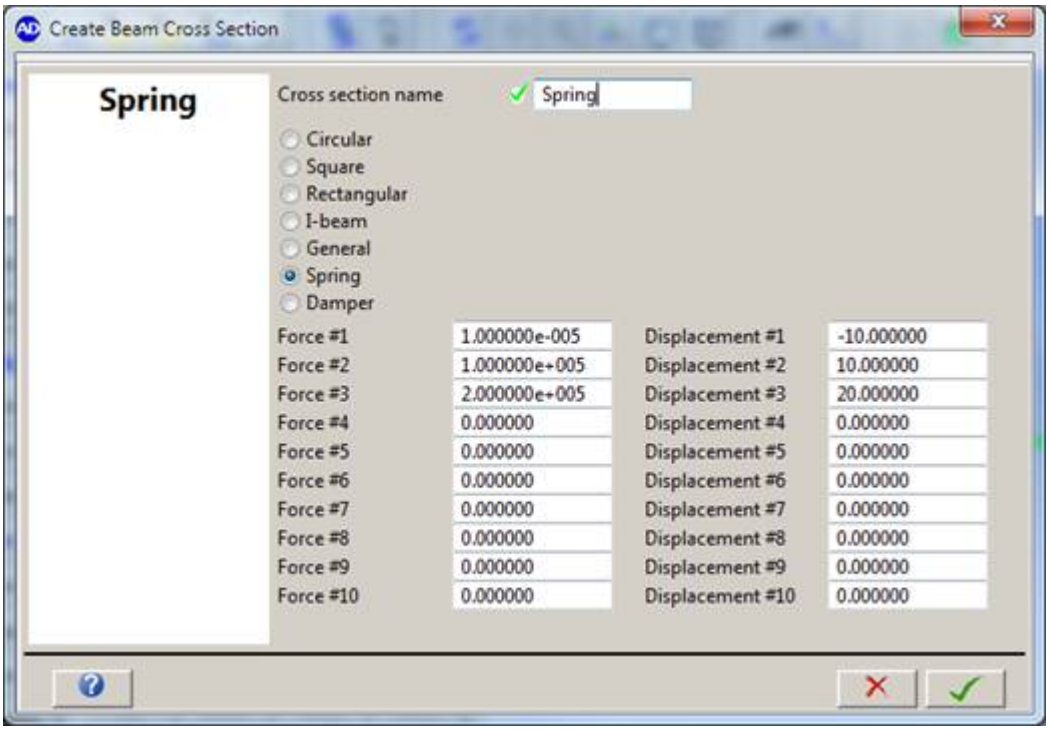
General – The cross section area, the 2nd moment of area about axis 22 and 33, the unit vector 22, and shear area along axis 22 and 33 need to be defined.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



10.10.6.2. Spring Cross Section

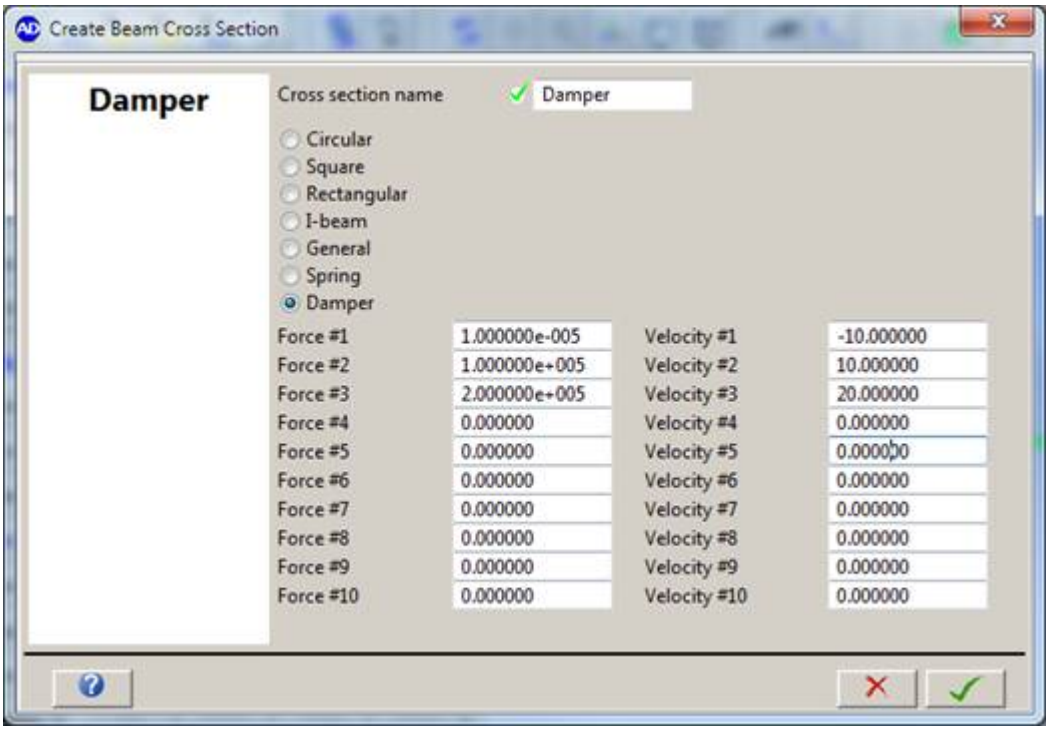
For a spring, the cross section needs to define a 10 point piecewise linear force-displacement behavior.



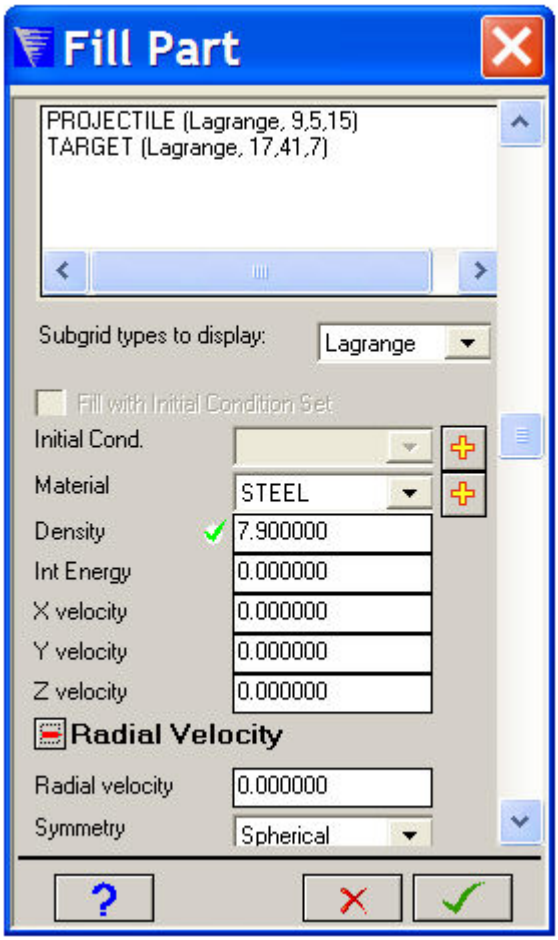
10.10.6.3. Damper cross section

For a damper, the cross section needs to define a 10 point piecewise linear force-velocity behavior.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



10.10.7. Parts - Fill - Multi-Fill



Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

This window lets you quickly fill multiple Parts. Each selected Part must be filled in its entirety.

Fill Parts

The top box displays a list of Parts that you can select to fill.

Below this box is a pull-down menu that lets you choose the type of Parts from which you want to make your selection (you can only fill Parts of the same type at the same time).

Fill with Initial Condition Set

Check this box if you want to fill the Parts using an Initial Condition Set.

The advantage of filling a Part using an Initial Condition Set is that if you later change one or more parameters in the set, you do not have to refill your Part.

Use the pull-down menu to select the Initial Condition Set you want to use.

You can define new Initial Condition Sets by clicking the **Init.Cond.** button on the Navigation Bar or by clicking next to the option in this window.

Material

If you do not want to use an Initial Condition Set, select the material you want to use to fill your Part from this Pull-down menu.

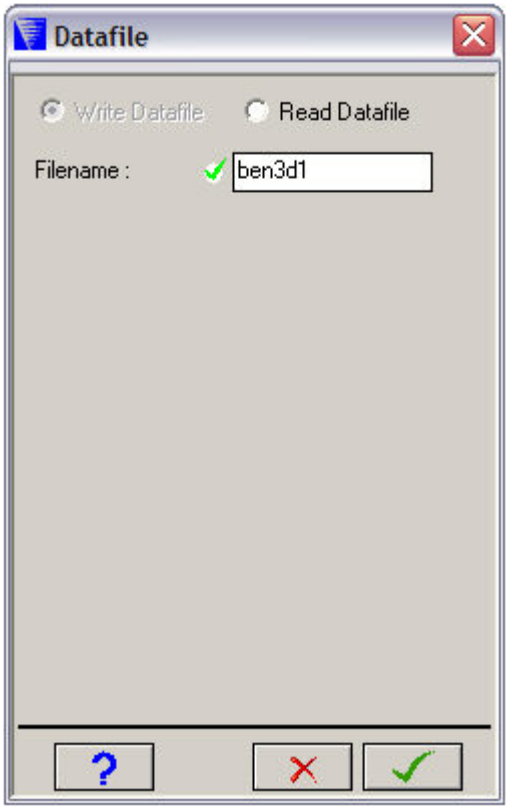
You can define new materials by clicking the **Materials** button on the Navigation Bar or by clicking next to the option in this window.

Parameters

If you do not want to use an Initial Condition Set, enter parameter values for your fill (**Density**, **Energy**, **Velocity**).

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

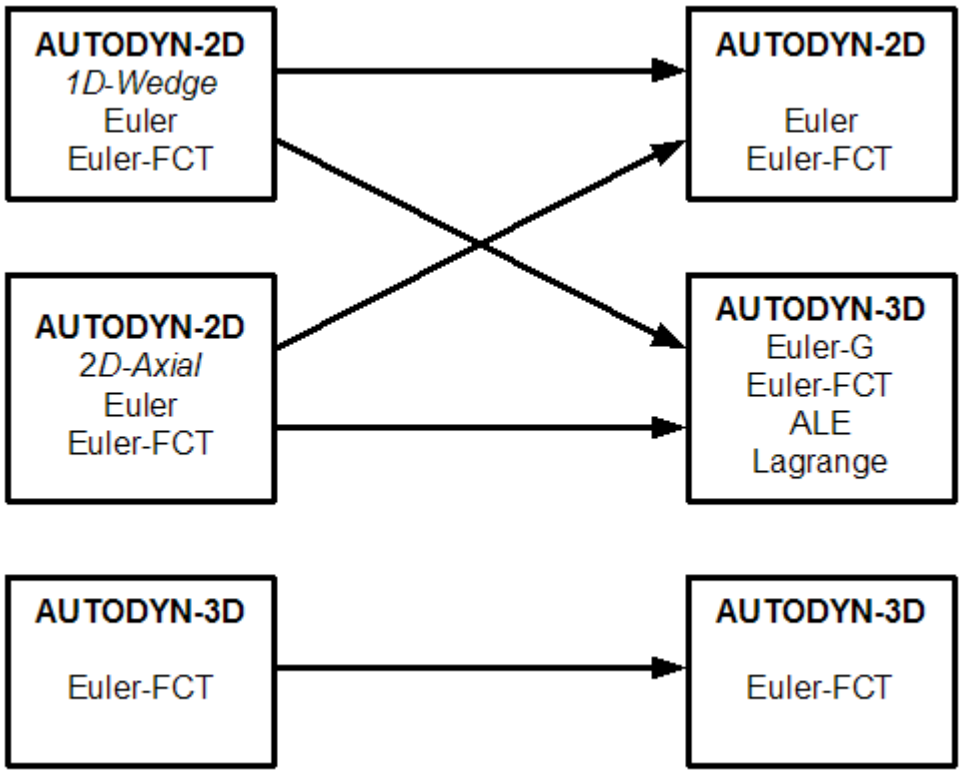
10.10.8. Parts - Fill - Datafile



This window lets you write fill data to or from datafiles (**.fil**).

These data files are commonly used for remapping solutions from 1D Wedge, 2D axysymmetric and 3D models into other 2D / 3D models. The figure below illustrates the remap operations available in Autodyn:

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



- The Autodyn-2D remap (data) file writing capability includes all state variables including the stress tensor etc. You retain the existing option to only write hydrodynamic data if you wish to keep the size of the remap files to a minimum.
- The Autodyn-2D remap (data) file import capability allows 2D to 2D remapping of all state variables including the stress tensor etc.
- The Autodyn-3D remap (data) file import capability includes all state variables including the stress tensor etc.
- In Autodyn-3D an approximate remap into single material ALE and Lagrange solvers has also been included, that is cells are filled with the material with the largest volume fraction. If this happens to be void material then the element is set to unused.
- When reading remap data files, the origin of the remap can be user defined in both Autodyn-2D and 3D. Furthermore, in Autodyn-3D the axis of symmetry for the remap can be aligned with either the X(I), Y(J) or Z(K) direction. This reduces the existing constraint that the X-axis needed to be consistent between the 2D and 3D models whilst keeping the geometry calculation simple.
- Note that all remap data files created in older versions of the software can still be read into the latest version Autodyn, although they will only contain hydrodynamic material data.
- The results of the remapping of the Johnson-Holmquist material model cannot be guaranteed to be accurate when the material is damaged considerably, since voids are artificially closed during remapping. You should therefore take care to check results carefully when using the Johnson-Holmquist material model in the remapping procedure.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Write Datafile

Select this option if you want to write fill data for the current Part to a **.fil** file.

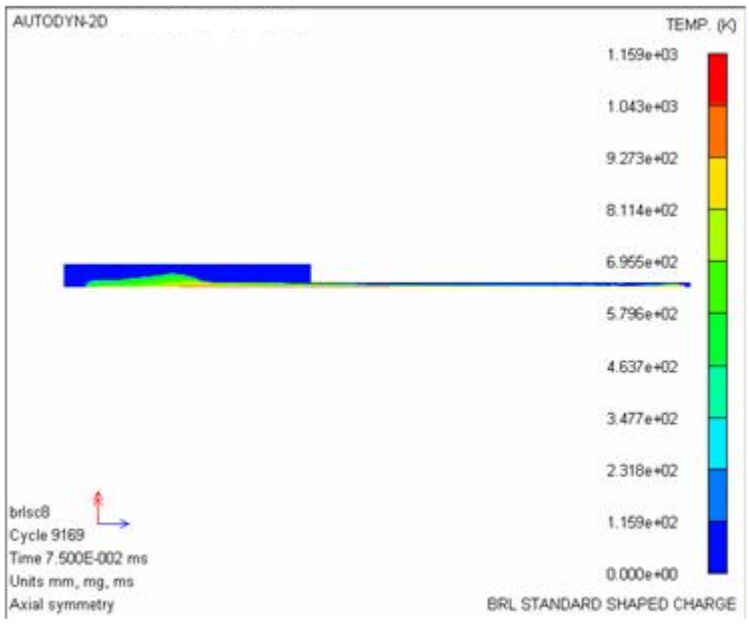
Read Datafile

Select this option if you want to read fill data for the current Part from a previously written **.fil** file.

Filename

The file name (do not append **.fil**)

An example of a shaped charge analysis is shown in the below. The formation of the shaped charge jet is first modeled using the multi-material Euler solver in Autodyn-2D, as shown below. A remap data file is created at the desired wrap-up time.

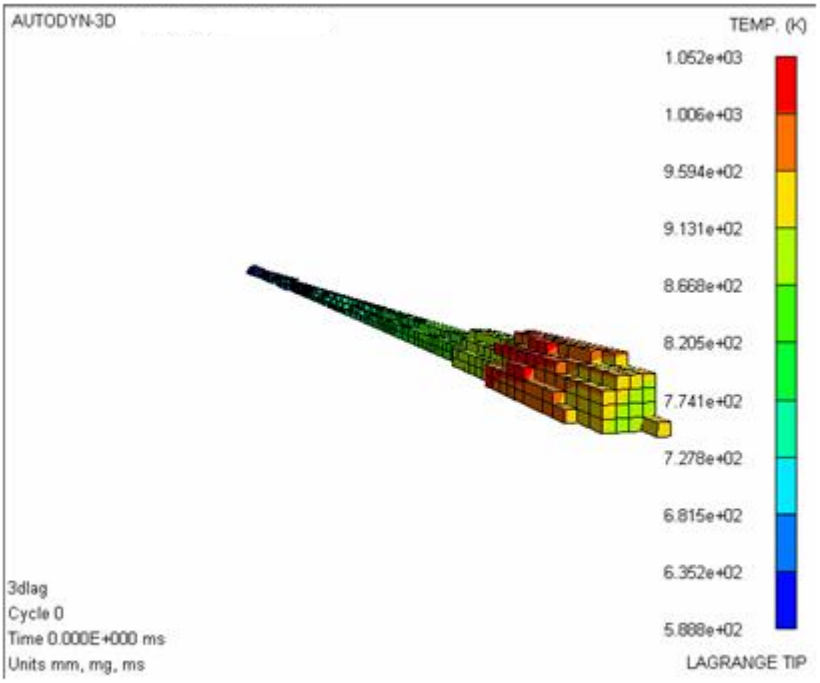


Formation of shape charge jet in AUTODYN-2D

The analysis can then be continued using Autodyn-3D. In this example a Lagrange part is first created and zoned in order that the part will completely enclose the volume occupied by the shaped charge jet. At this stage the part is not filled and all cells are left as unused. Note that it is recommended that cell sizes are chosen such that at least 4 filled elements will exist across the radius of the shaped charge jet. These cell sizes can be determined by examining the geometry at the wrap-up cycle of the Autodyn-2D analysis.

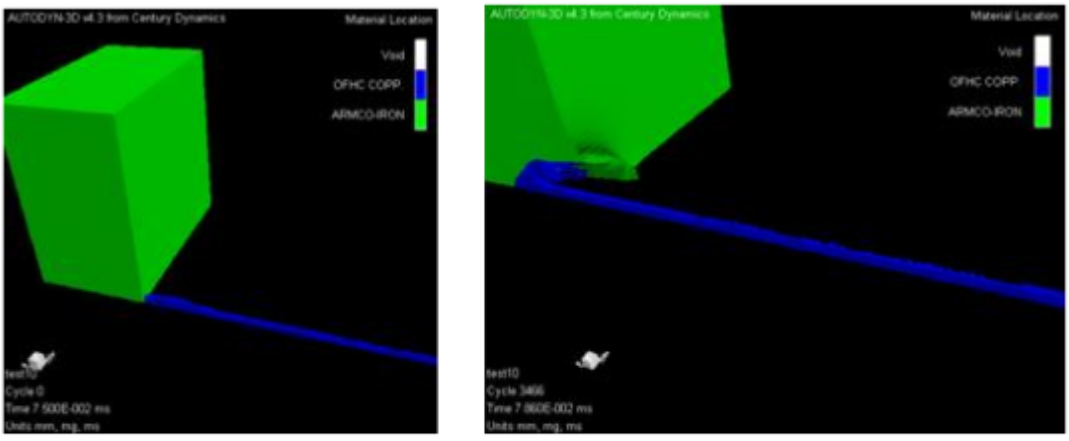
In Autodyn-3D the results from the 2D multi-material Euler analysis are remapped into the empty 3D Lagrangian part. The Lagrangian cells that contain more than 50% material following the remap are set as used cells. The profile of the Lagrangian jet will therefore not exactly match that of the 2D case, although accuracy can be enhanced by using the greatest level of refinement that computation time constraints will allow. Note that the constitutive state variables and velocities for the used cells are also remapped into the 3D analysis. An example of the tip of a shaped charge jet following the remap procedure described above is shown below.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Lagrangian jet following remap into AUTODYN-3D

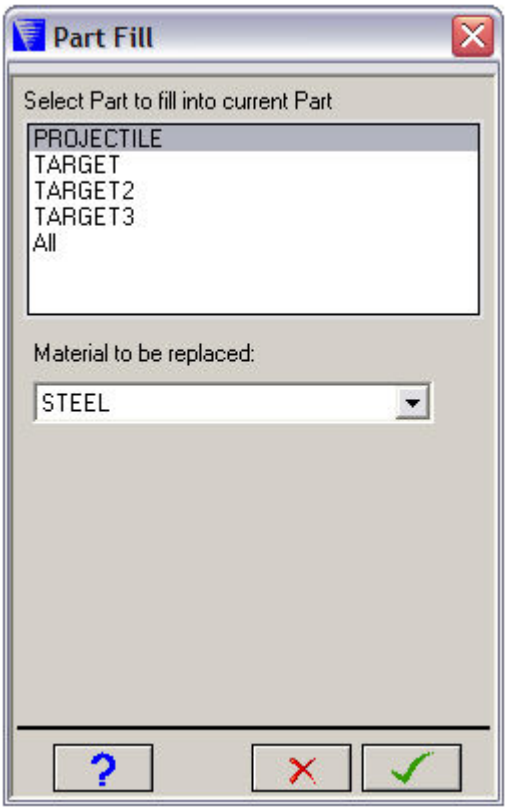
The 3D Lagrangian part can then be translated or rotated and used in several analyses including normal and oblique impacts onto several different targets. The following figures show an example in which a Lagrangian target has been introduced into the analysis and the penetration of the jet into the target has been simulated. Note that erosion models were included for both the jet and target materials.



Normal impact onto Lagrangian target in AUTODYN-3D
(a) initial configuration (b) shape charge penetrates target

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

10.10.9. Parts - Fill - Part Fill



This window lets you fill an Euler Part by mapping the materials and initial conditions of an existing structured **Lagrange** or **Fill Part** or unstructured volume-element parts into the Euler Part.

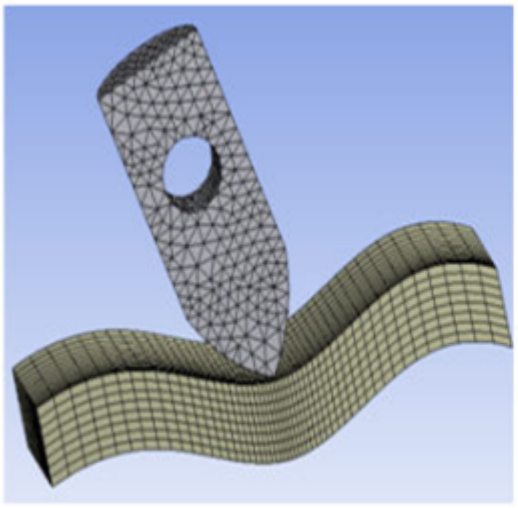
Select Part to fill into current Part

The Lagrange or Fill Part you want to use to fill your current Euler Part.

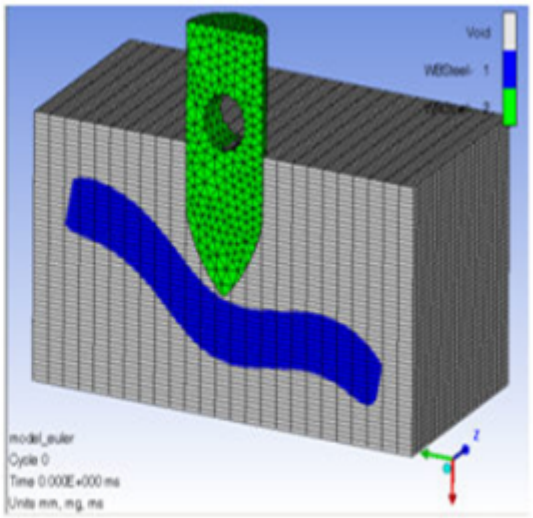
Material to be replaced

The material to be replaced during the fill operation (this information is used only if Euler cells contain multiple materials).

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

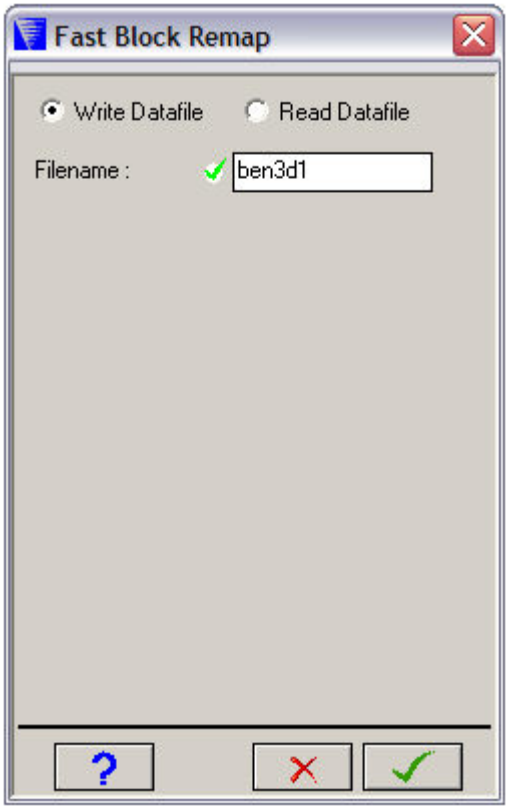


Mesh of Projectile and Target



Multi-material mesh filled with Unstructured Part

10.10.10. Parts - Fill - Block Remap



This window lets you write fill data to or from datafiles (**.fil**) for performing fast block remaps.

The standard remapping scheme for 3D to 3D Euler-FCT is very useful in blast modeling but in certain circumstances can take a very long time to search for the cells in the remapping range, especially if the part size is very large, for example more than 2 million cells.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

The "fast block" remapping command maps several cells from a fine mesh of a small space into a single cell in a relatively coarse mesh of a large space, directly without searching for the remapping range. Thus the efficiency of the remapping process has been improved dramatically.

The fast block remapping procedure is applicable to Euler-FCT parts only. The scheme allows unused cells in the remapping region. However, cover fractions are not considered in the remapping calculations so block remapping should not be applied to any region of the part where Euler-FCT/Lagrange or Euler-FCT/Shell coupling takes place.

Write Datafile

Select this option if you want to write fill data for the current Part to a **.fil** file.

Read Datafile

Select this option if you want to read fill data for the current Part from a previously written **.fil** file.

Filename

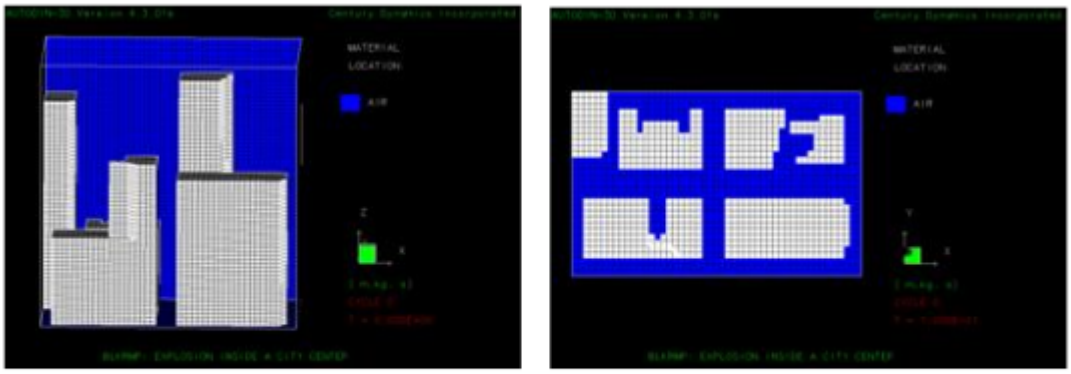
The file name (do not append **.fil**)

Typical steps that will be undertaken when using Block Remapping for blast problems are as follows:

1. Load the original building data file.
2. Modify the part so it only contains a small area of the explosive and the surrounding buildings.
3. Refine the mesh, then remap 1D or 2D data to the refined part.
4. Save the file and start calculation.
5. At the wrap-up cycle (when the explosive wave approaches the boundary of the refined part), output the blast field to a remap file.
6. Load the original building data file again.
7. Modify the part so it contains a larger area.
8. Refine the mesh.
9. Read the remap file.
10. Save the file and continue running.
11. Repeat 5-10 as necessary.

An example of an analysis of a blast in a city center that was performed using the block remapping facility is shown below. The city center is represented using an Euler-FCT mesh with unused regions to represent buildings as shown below. Note that the original zoning and fill operations for this model were set up using the new block fill command.

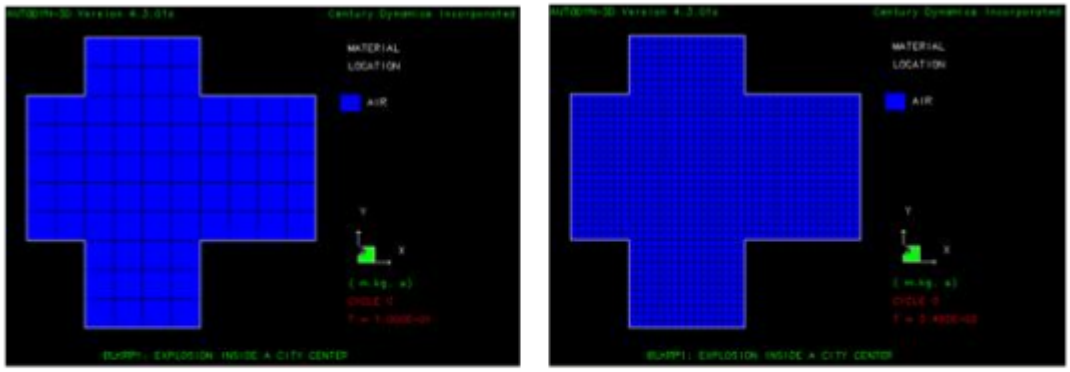
Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



(a) side view (b) plan view

Layout of original Euler-FCT part representing city centre

To perform the initial expansion of the blast the central section of the grid, as shown below in figure a, is taken and refined by a factor of 4 as shown in figure b.



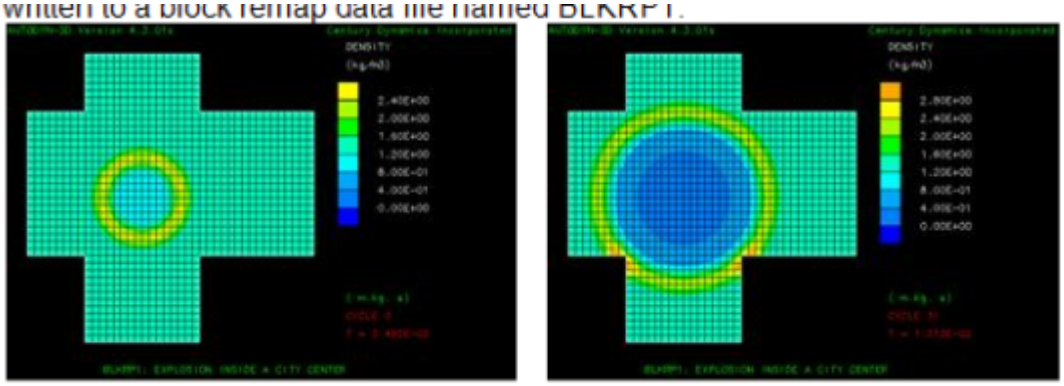
(a) unrefined (b) refined by factor of 4x4x4

Central section of part (Range I=21-31, J=11-21, K=1-21)

The analysis of the detonation of the spherical charge is initially performed using a 1D wedge analysis which is then remapped into the refined grid as shown below. The expansion of the blast wave in the central region of the city is then performed using the refined 3D Euler-FCT grid until the wave front reaches the boundaries of the refined region as shown in figure b. At this point the results for this analysis are written to a block remap data file named BLKRP1.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

written to a block remap data file named BLKRP1.

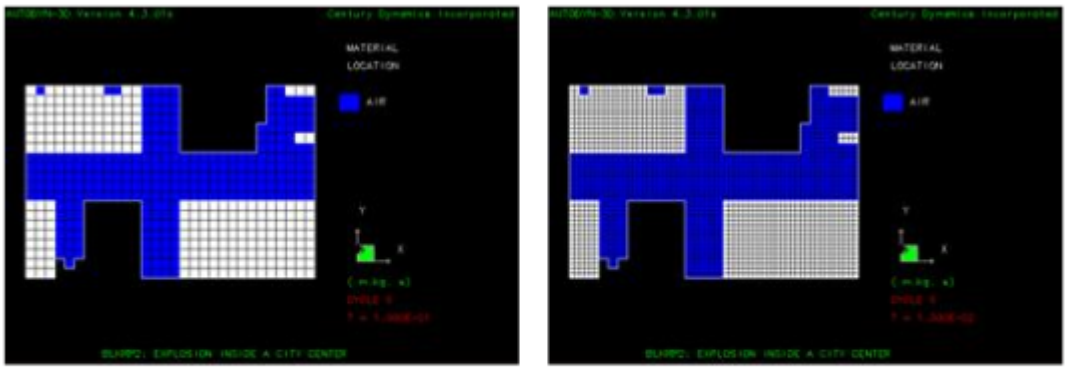


(a) (b)

(a) initial blast remap from 1D wedge data
(b) blast wave reaches boundary of refined region

Expansion of blast in refined central region

Following the creation of the remap data file BLKRP1 the original model is subsequently reloaded and a larger region of the Euler-FCT part is analyzed as shown below in figure a. This section is refined by a factor of 2 in all directions to give the resulting mesh shown in figure b.

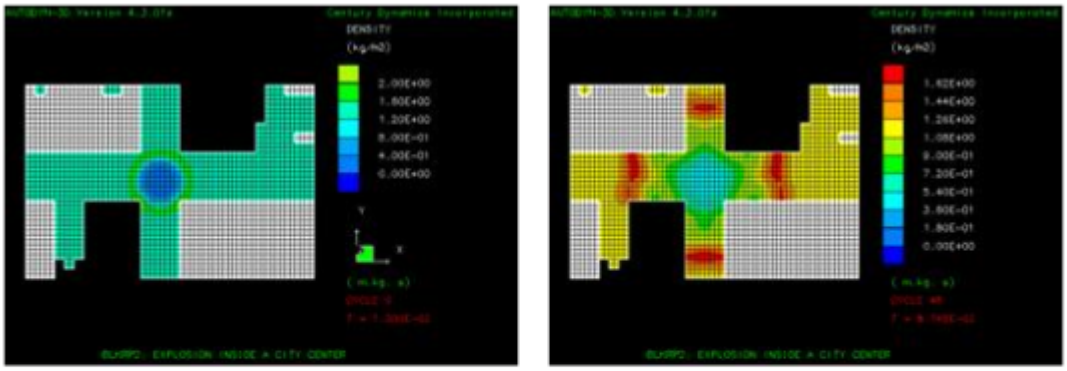


(a) unrefined (b) refined by factor of 2x2x2

Larger central section of part (Range I=11-41, J=6-26, K=1-41)

The results stored in file BLKRP1 are then remapped into the refined grid to give the initial conditions for the second phase of the analysis as shown below in figure a. The propagation of the blast wave can then continue until again the wave front approaches the boundaries of the refined region as shown in figure b. At this stage a further block remap data file is written named BLKRP2.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

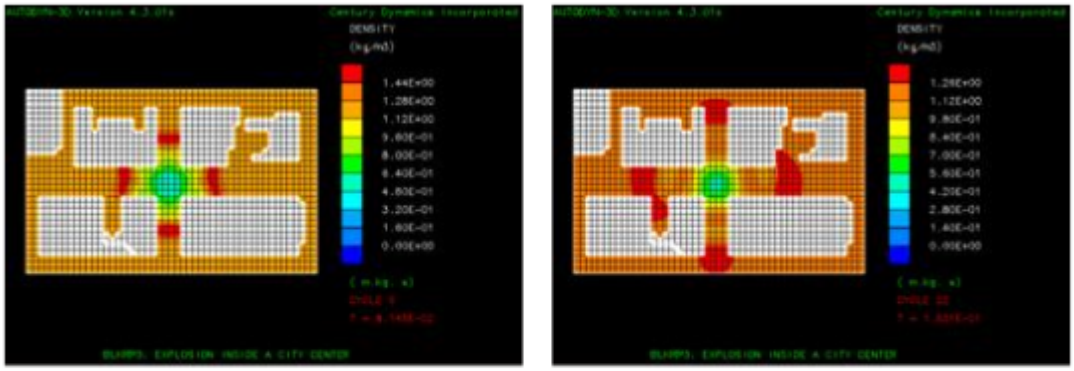


(a) (b)

(a) initial blast remap from BLKRP1
(b) blast wave approaches boundary of refined region

Expansion of blast in large refined region

The final stage of the analysis can be performed using the full model. This model is reloaded and the results stored in file BLKRP2 are remapped into the entire Euler-FCT part to give the initial conditions for the final phase of the analysis as shown below in figure a. The propagation of the blast wave can then be tracked though the remainder of the city center as shown in figure b.



(a) (b)

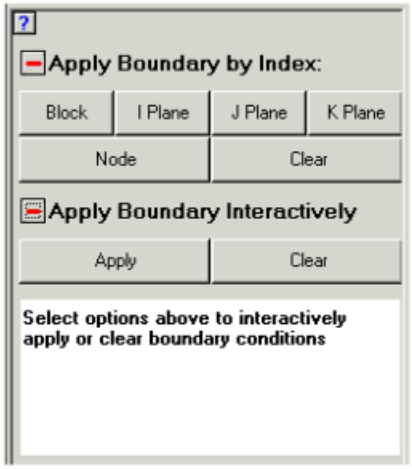
(a) second blast remap from BLKRP2
(b) blast wave propagates through city centre

Expansion of blast in full Euler-FCT part

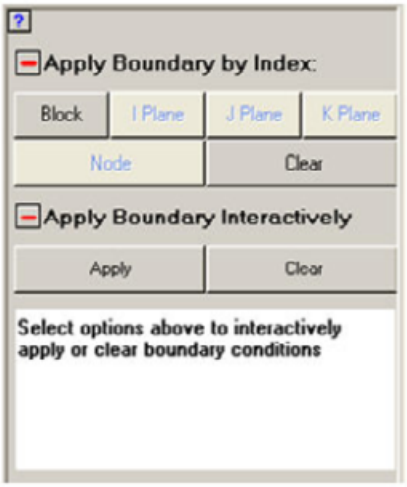
10.11. Parts - Boundary

Clicking the **Boundary** button in the **Parts panel** displays the following dialog in the lower region of the panel:

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Structured



Unstructured

This dialog lets you apply boundary conditions to the selected Part.

Block

Apply a boundary condition to a block in index space.

I/J/K Plane

Apply a boundary condition to a plane in index space.

Node

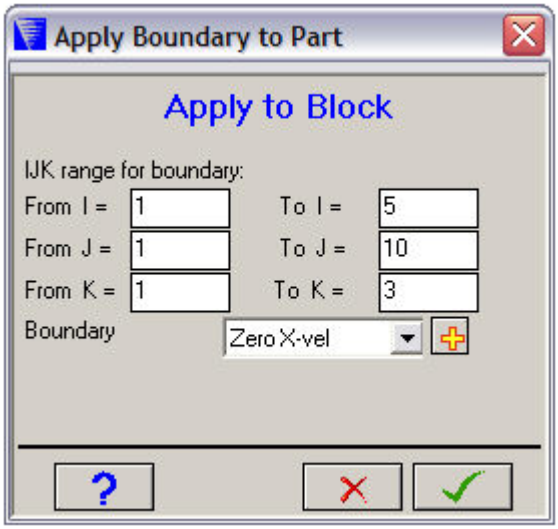
Apply a boundary condition to a node.

Clear

Clear previously applied boundary conditions.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

10.11.1. Parts - Boundary - Block



This window lets you apply a boundary condition to a block in index space of the current structured Part.

For an unstructured Part the boundary condition will be applied to all nodes, elements or faces of a Part.

You cannot apply two different Velocity boundary conditions to structured beam parts.

IJK range for boundary

The IJK range of the block.

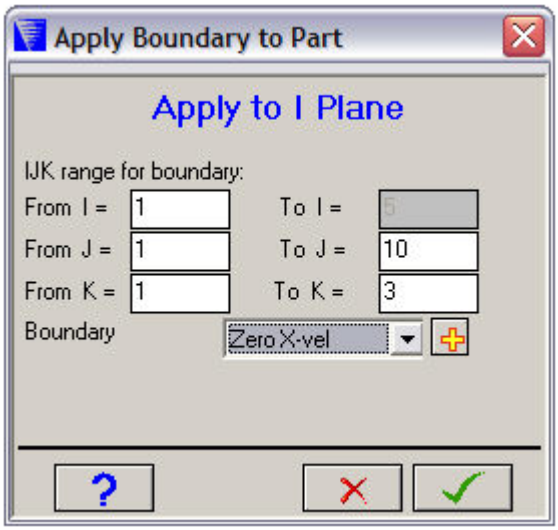
Boundary

Select the boundary condition you want to apply from this pull-down menu.

You can define Boundary Conditions by clicking the **Boundaries** button in the Navigation Bar, or you can click next to the pull-down menu.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

10.11.2. Parts - Boundary - I / J / K Plane



This window lets you apply a Boundary Condition to an I, J, or K plane of the current Part.

The example shown is for applying a Boundary Condition to an I plane (input for the other planes is similar).

You cannot apply two different Velocity boundary conditions to structured beam parts.

IJK range for boundary

The I plane and the J and K range.

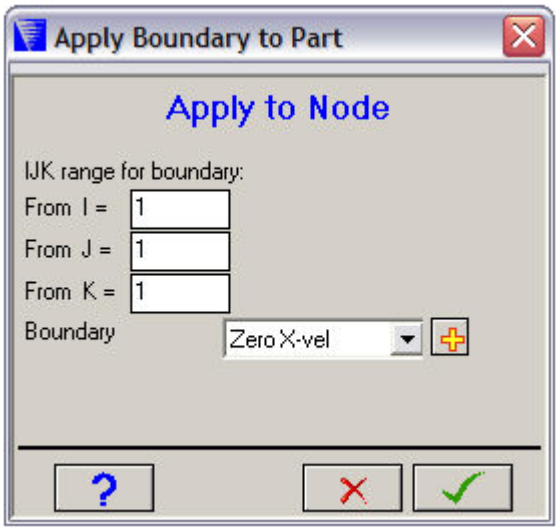
Boundary

Select the boundary condition you want to apply from this pull-down menu.

You can define Boundary Conditions by clicking the **Boundaries** button in the Navigation Bar, or you can click next to the pull-down menu.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

10.11.3. Parts - Boundary - Node



This window lets you apply a Boundary Condition to a node.

You cannot apply two different Velocity boundary conditions to structured beam parts.

IJK range for boundary

The I, J, and K index for the node.

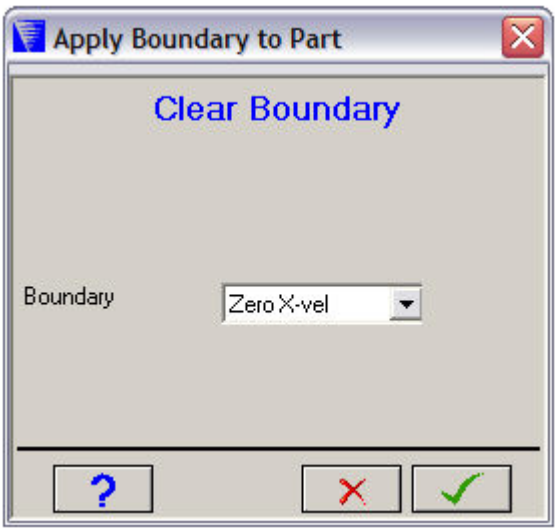
Boundary

Select the boundary condition you want to apply from this pull-down menu.

You can define Boundary Conditions by clicking the **Boundaries** button in the Navigation Bar, or you can click next to the pull-down menu.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

10.11.4. Parts - Boundary - Clear

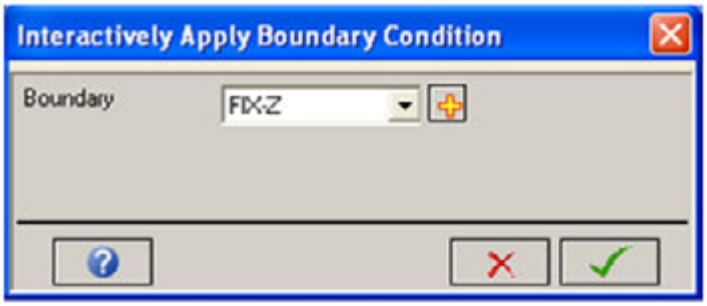


This window lets you clear a Boundary Condition from the current Part.

This operation does not delete the definition of the Boundary Condition, so you can reapply it later.

Select the boundary condition you want to clear from the pull-down menu and click .

10.11.5. Parts - Boundary - Apply Interactively



This window lets you apply a boundary condition interactively by creating a 2D polygon in the graphics display. The boundary condition is applied to any nodes or faces that lie within the 2D polygon.

Interactive selection (Apply/Clear) of boundaries is not available when the Shell Thickness plot is active.

Boundary

Select the boundary condition you want to apply from this pull-down menu.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

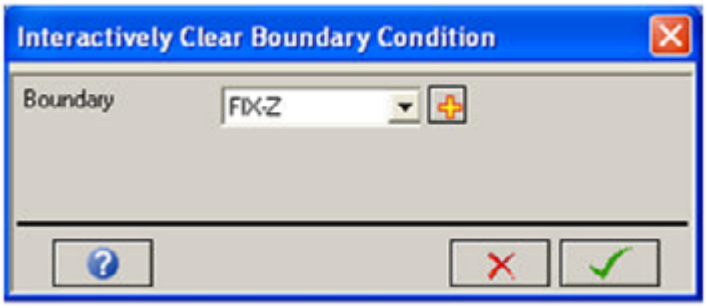
You can define Boundary Conditions by clicking the **Boundaries** button in the Navigation Bar, or you can click next to the pull-down menu.

Close the window with the button, to enter the polygon definition stage.

Select the polygon points using **Alt**+left mouse button. To delete the last polygon, use **Shift**+left mouse button. To complete the polygon, use **Ctrl**+left mouse button.

After completing the polygon, the selected nodes or faces are displayed. Click the button to accept this selection, and apply the boundary condition. At any stage, the button can be used to cancel the process.

10.11.6. Parts - Bound - Clear Interactively



This window lets you clear a boundary condition interactively by creating a 2D polygon in the graphics display. The boundary condition is removed from any nodes or faces that lie within the 2D polygon.

Interactive selection (Apply/Clear) of boundaries is not available when the Shell Thickness plot is active.

Boundary

Select the boundary condition you want to apply from this pull-down menu.

Close the window with the button, to enter the polygon definition stage.

Select the polygon using **Alt**+left mouse button. To delete the last polygon point, use **Shift**+left mouse button. To complete the polygon, use **Ctrl**+left mouse button.

After completing the polygon, the selected nodes or faces are displayed. Click the button to accept this selection, and apply the boundary condition. At any stage, the button can be used to cancel the process.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

10.12. Parts - Gauges

Clicking the **Gauges** button in the **Parts panel** displays the following dialog in the lower region of the panel :

This dialog lets you define gauge points in the current Part.

Add

Add one or more gauge points.

Move

Relocate a gauge point.

Delete

Delete one or more gauge points.

Review

Review all your gauge points in a window.

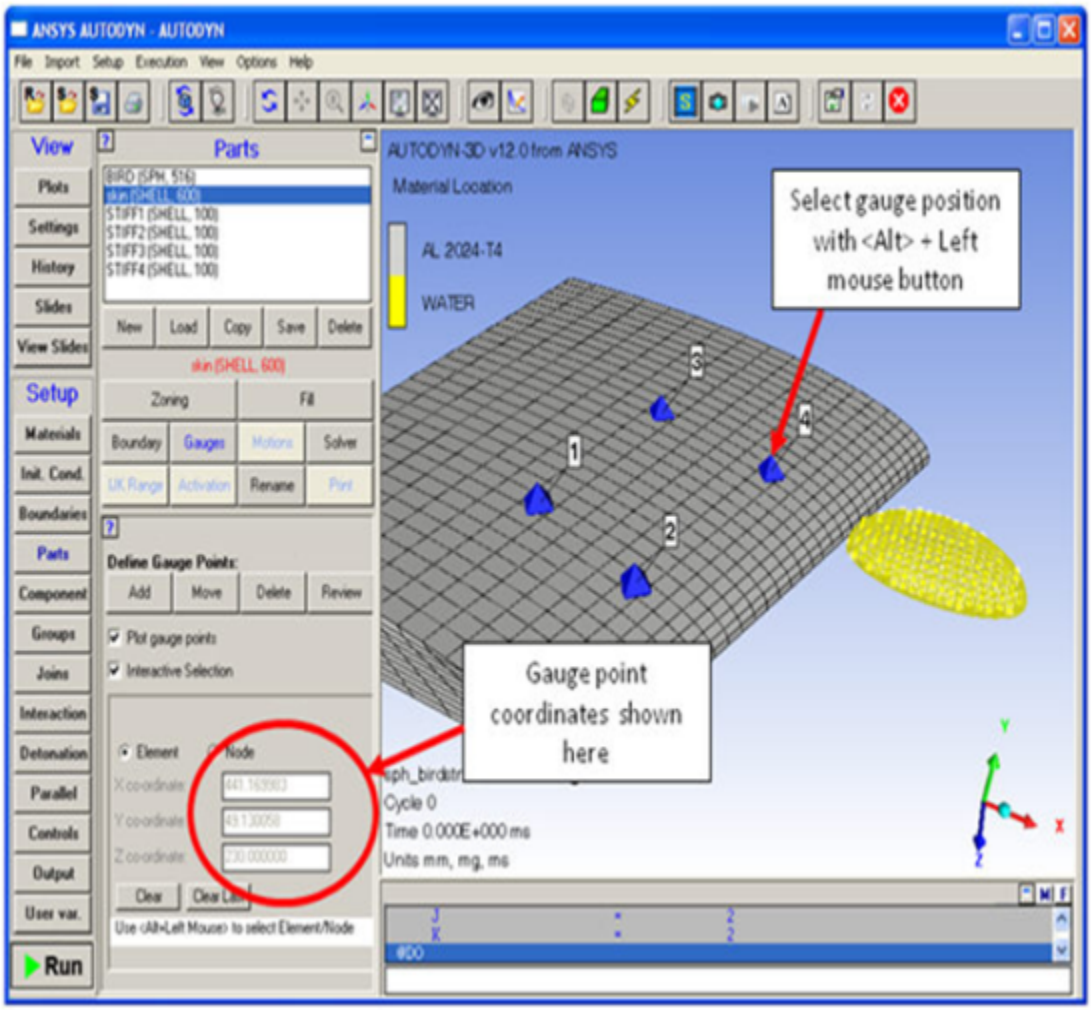
Plot gauge points

Check this box to see the location of you gauge points in the view panel.

Interactive Selection

Use the mouse to interactively select Elements or Nodes where you wish to place your gauge points by switching on Interactive Gauge points in the Gauges Panel.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



The IJK indices or XYZ position of the selected element/node will be displayed, as they are selected.

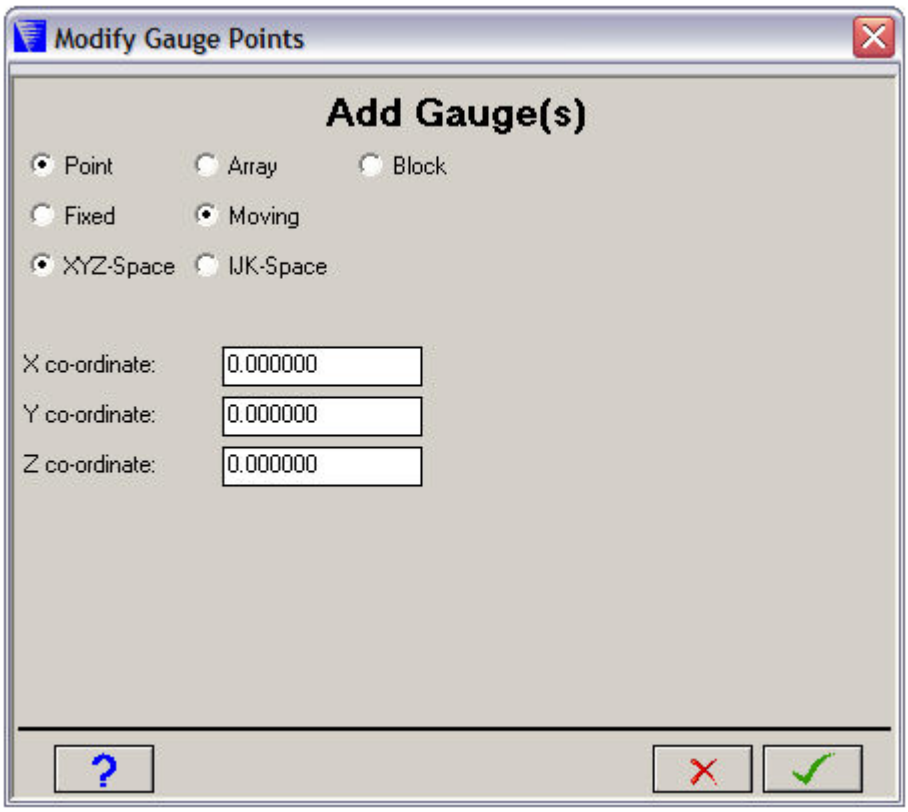
Gauges added via this method will also be cached until cleared by selecting the Undo button, or the Gauges Panel is closed.

When adding Gauges via this method the part containing the Element or Node selected becomes the selected Part in the Parts panel.

The interactive selection of Gauge Points is not available in examine mode, and the panel will not be visible if this is the case.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

10.12.1. Parts - Gauges - Add



This window lets you add gauges to your model.

A gauge is a location where time histories of selected variables are recorded during a calculation.

Point / Array / Block

Select whether you want to define a single gauge (**Point**), a 2D array of gauges (**Array**) or a 3D block of gauges (**Block**).

Fixed / Moving

Select whether you want the gauge(s) to be fixed in space or move with the flow (as if imbedded in the material). Moving gauges cannot be defined in Euler subgrids in 3D analyses. Fixed gauges should not be used in structured Lagrange parts when running in parallel.

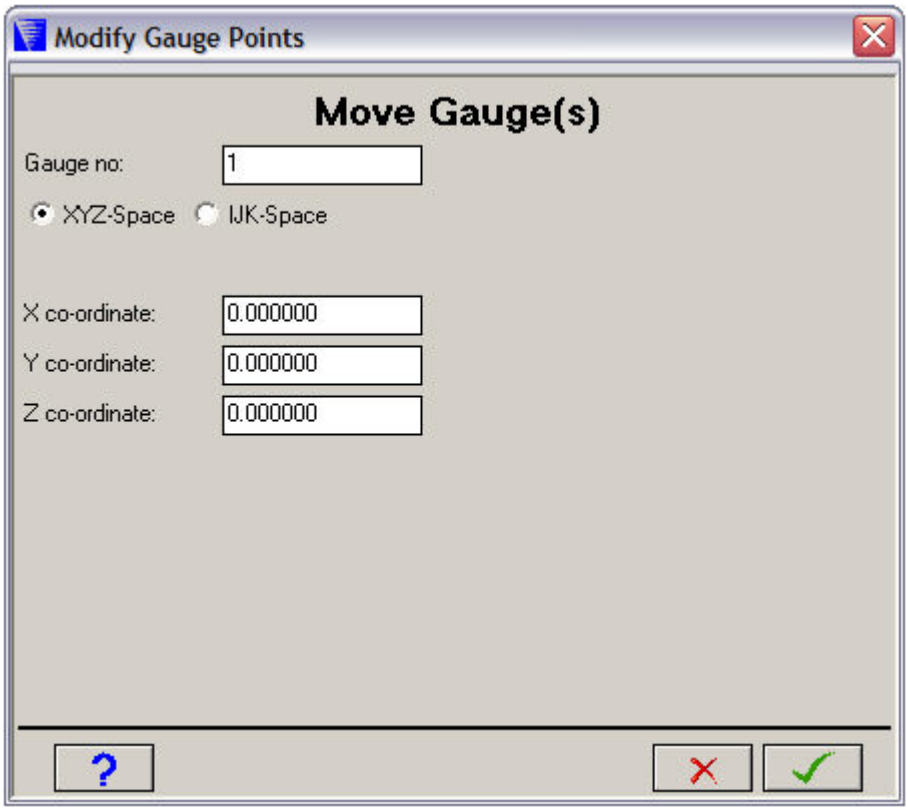
XYZ / IJK Space

Select whether you want to specify your gauge locations in real space (XYZ) or index space (IJK)

Depending on your selections, the remaining fields will ask you to specify the (X,Y,Z) coordinates of your gauges or the (I,J,K) indices of the elements in which your gauges are to be placed.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

10.12.2. Parts - Gauge - Move



This window lets you relocate one gauge point.

Gauge Number

The number of the gauge you want to relocate (gauges are assigned numbers in the order in which you define them).

XYZ / IJK Space

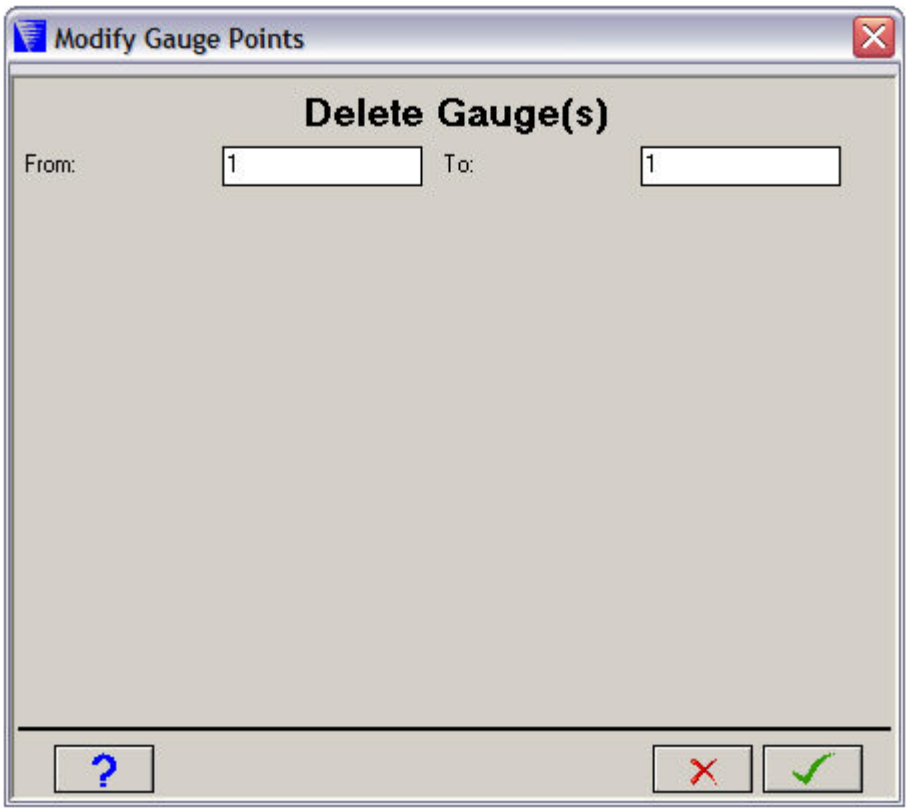
Select whether you want to specify your gauge locations in real space (XYZ) or index space (IJK)

New Position

The (X,Y,Z) coordinates or (I,J,K) indices for the new position of the gauge.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

10.12.3. Parts - Gauge - Delete

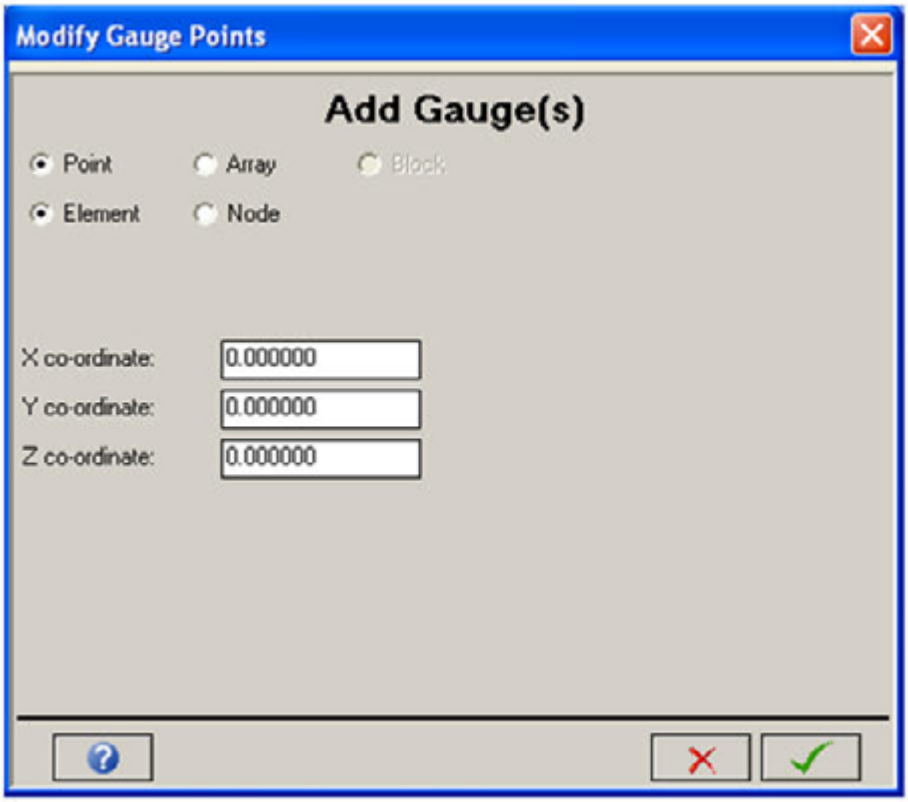


This window lets you delete one or more gauges.

Enter the range of the gauge numbers you want to delete and click .

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

10.12.4. Parts - Gauges - Add Unstructured



This window lets you add gauges to an unstructured part.

A gauge is a location where time histories of selected variables are recorded during a calculation.

Point/Array

Select whether you want to define a single gauge (Point) or a 2D array of gauges (Array).

Element/Node

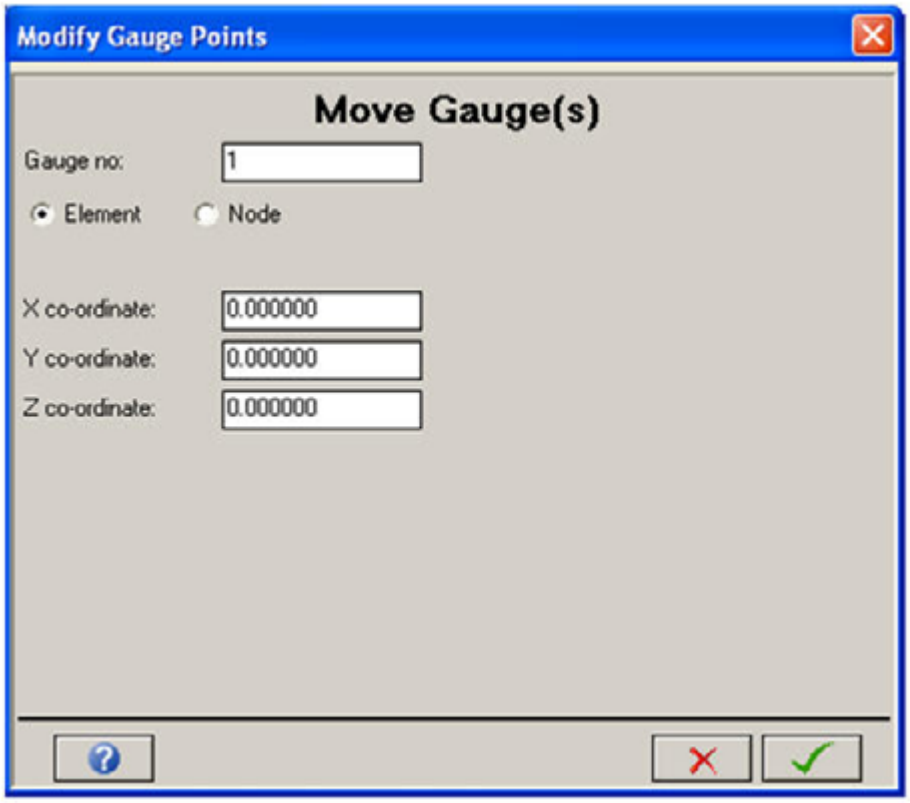
Select whether you want to associate gauge points with element or nodes.

Coordinates

The remaining fields will ask you to specify the (X, Y, Z) coordinates of your gauges.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

10.12.5. Parts - Gauge - Move Unstructured



This window lets you relocate one gauge point.

Gauge Number

The number of the gauge you want to relocate (gauges are assigned numbers in the order in which you define them).

Element/Node

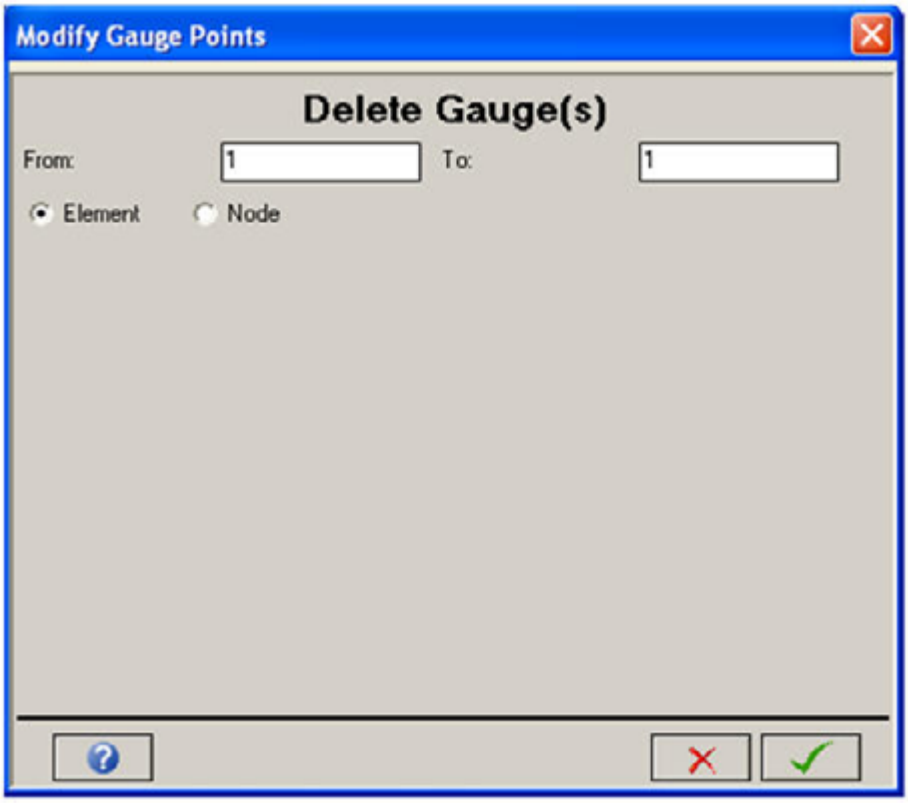
Select whether you want to associate gauge points with element or nodes.

New Position

The (X, Y, Z) coordinates for the new position of the gauge.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

10.12.6. Parts - Gauge - Delete Unstructured



This window lets you delete one or more gauges.

Enter the range of the gauge numbers you want to delete and click .

10.13. Parts - Solvers (Structured)

Clicking the **Solvers** button in the **Parts panel** displays the following dialog in the lower region of the panel:

This dialog lets you change some solver settings for the current Part.

Type

Change the type of solver used.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Solution

Change the solution type associated with some solvers.

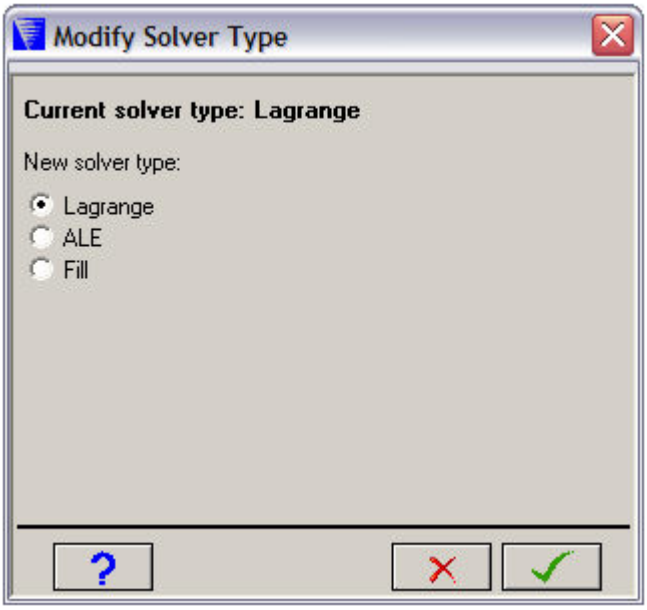
Dezone

Perform dezoning for 2D Lagrange and Euler solvers and for 3D Lagrange, ALE, and Euler solvers.

Jetting

Add the Jetting option to a 2D shell.

10.13.1. Parts - Solvers (Structured) - Type



This window lets you change the type of solver assigned to the current Part.

You can change only to a solver compatible with the current solver used for the Part.

Current solver type

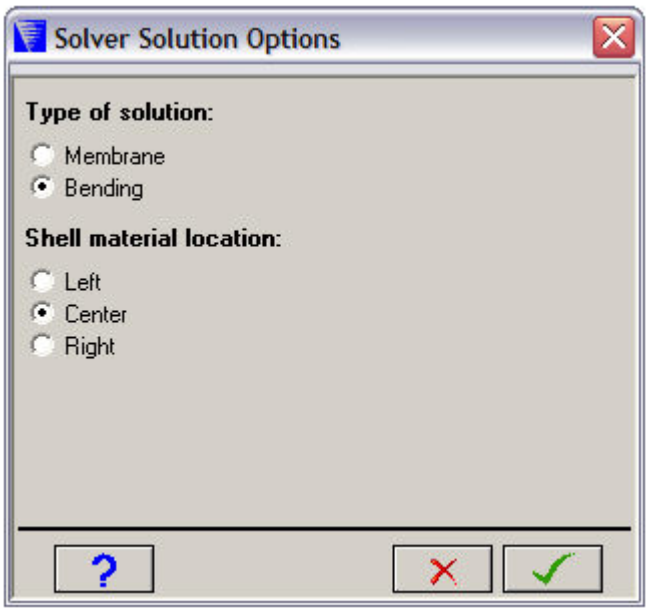
The current solver type.

New solver type

A list of the solver types compatible with the current solver type (if any). Select the solver type you want to use from this list.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

10.13.2. Parts - Solvers (Structured) - Solution (2D Shells)



This window lets you set solution options for 2D shell Parts.

Type of Solution

Select whether you want to compute a **membrane** solution or a solution with full **bending** moments.

Shell material location

Shell segments have no geometrical thickness, so this option lets you select the location of the shell material relative to its geometric position.

Left

Material is located to the left of the shell as you progress from the first node to the last node.

Center

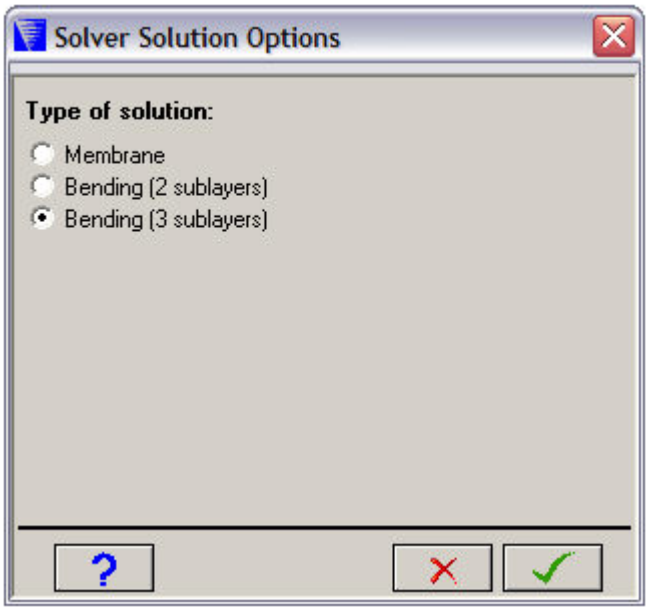
Material is located equally on either side of the shell (that is, from -h/2 to h/2, where h is the thickness).

Right

Material is located to the right of the shell as you progress from the first node to the last node.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

10.13.3. Parts - Solvers (Structured) - Solution (3D Shells)

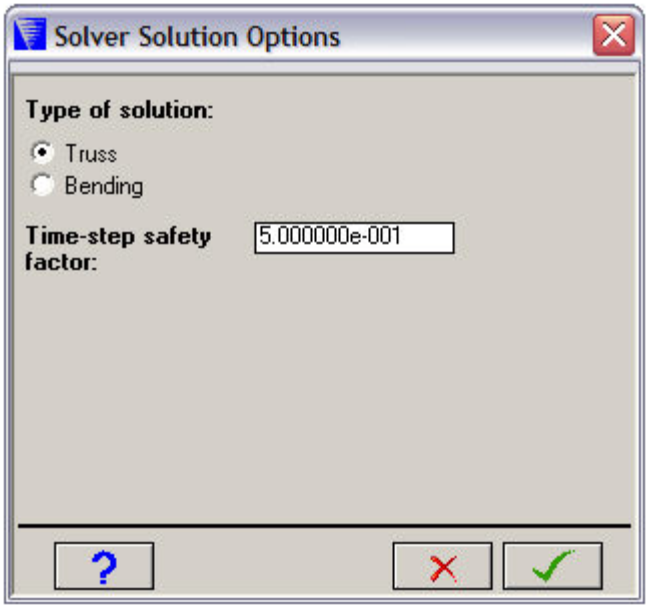


This window lets you set solution options for 3D shell Parts.

Type of Solution

Select whether you want to compute a **membrane** solution or a solution with full **bending** moments integrating over 2 or 3 sublayers.

10.13.4. Parts - Solvers (Structured) - Solution (Beams)



This window lets you set solution options for Beam Parts.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

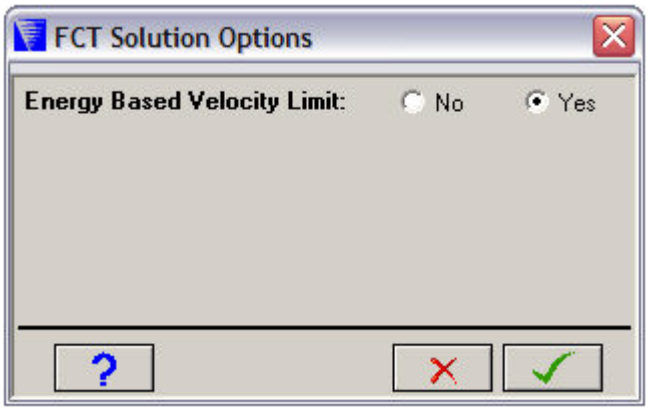
Type of Solution

Select whether you want to compute a **truss** solution or a solution with full **bending** moments.

Time-step safety factor

The safety factor you want to apply to the time-step calculation. The default value usually ensures stability of your calculation.

10.13.5. Parts - Solvers (Structured) - Solution (3D Euler-FCT)



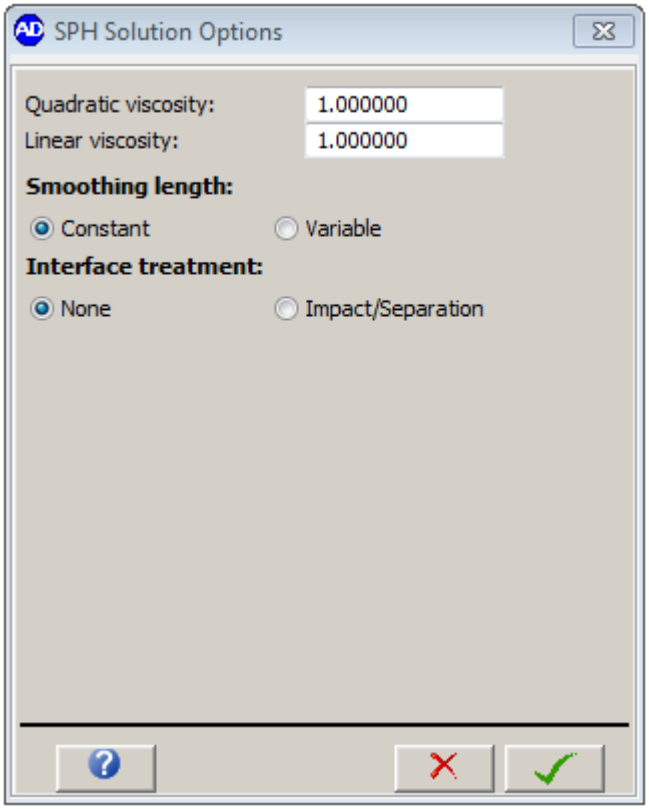
This window lets you set solution options for 3D Euler-FCT Parts.

Energy Based Velocity Limit

Specify whether or not you want to use an energy-based velocity limit for Euler-FCT computations.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

10.13.6. Parts - Solvers (Structured) - Solution (2D SPH)



This window lets you set solution options for 2D SPH Parts.

Quadratic viscosity

The coefficient of quadratic viscosity you want to use for SPH computations.

Linear viscosity

The coefficient of linear viscosity you want to use for SPH computations.

Smoothing Length

Specify whether you want to use a **constant** or **variable** smoothing length. Using a variable smoothing length is not recommended for most calculations.

Interface treatment

Specify if you want to use **Impact/Separation** calculations for the treatment of interfaces between different SPH regions.

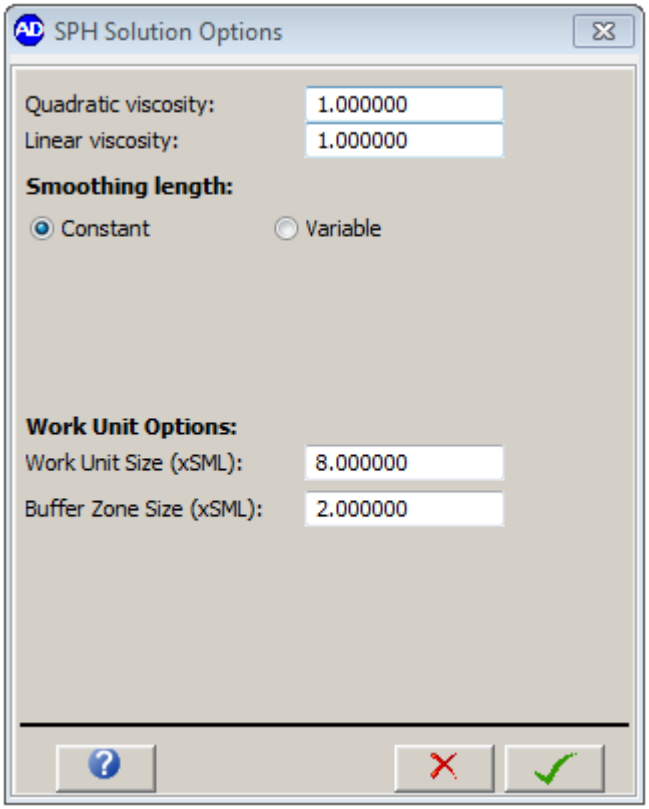
Note:

The current implementation of the Smooth Particle Hydrodynamics solver has the following limitation for Rigid Body Rotations:

- The conservation of angular momentum can not always be guaranteed.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

10.13.7. Parts - Solvers (Structured) - Solution (3D SPH)



This window lets you set solution options for 3D SPH Parts. The default settings are the recommended settings.

Quadratic viscosity

The coefficient of quadratic viscosity you want to use for SPH computations.

Linear viscosity

The coefficient of linear viscosity you want to use for SPH computations.

Smoothing Length

Specify whether you want to use a **constant** or **variable** smoothing length. Using a variable smoothing length is not recommended for most calculations.

Interface treatment

Specify if you want to use **Impact/Separation** calculations for the treatment of interfaces between different SPH regions.

Work Unit Options

Work Unit Size

The work unit size (as a multiple of the smoothing length) you want to use for SPH calculations. The value you enter here can significantly affect the efficiency of SPH calculations.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Buffer Zone Size

The buffer zone size (as a multiple of the smoothing length) you want to use for SPH calculations. The value you enter here can significantly affect the efficiency of SPH calculations.

Note:

The current implementation of the Smooth Particle Hydrodynamics solver has the following limitation for Rigid Body Rotations:

- The conservation of angular momentum can not always be guaranteed.

10.13.8. Parts - Solvers (Structured) - Dezone (2D & 3D Euler)

This window lets you dezone a 2D or 3D Euler (Ideal gas) or Euler (Multi-material) Part.

Dezoning involves combining a number of cells in a fine grid into single cells of a coarser grid.

I-factor

The number of cells in the I direction you want to combine into a single cell. This total number of cells in the I direction (IMAX-1) must be exactly divisible by this number.

J-factor

The number of cells in the J direction you want to combine into a single cell. This total number of cells in the J direction (JMAX-1) must be exactly divisible by this number.

Update grid size?

Select **Yes** if you want the IJ index range of the Part to be reduced automatically to reflect the dezone. If you select **No**, the index range will remain unchanged, with the higher indexed cells becoming unused.

K-factor (3D only)

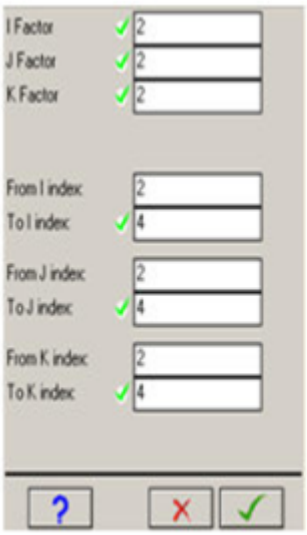
The number of cells in the K direction you want to combine into a single cell. This total number of cells in the K direction (KMAX-1) must be exactly divisible by this number.

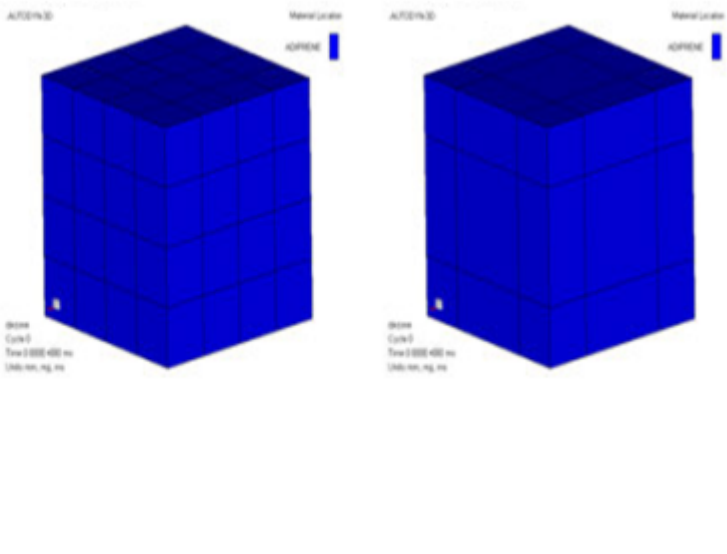
Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

10.13.9. Parts - Solvers (Structured) - Dezone (2D & 3D Lagrange/ALE)

This window lets you dezone a 2D or 3D Lagrange or ALE Part.

Dezoning involves combining a number of cells in a fine grid into single cells of a coarser grid.





I-factor

The number of cells in the I direction you want to combine into a single cell. This total number of cells in the I direction (IMAX-1) must be exactly divisible by this number.

J-factor

The number of cells in the J direction you want to combine into a single cell. This total number of cells in the J direction (JMAX-1) must be exactly divisible by this number.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

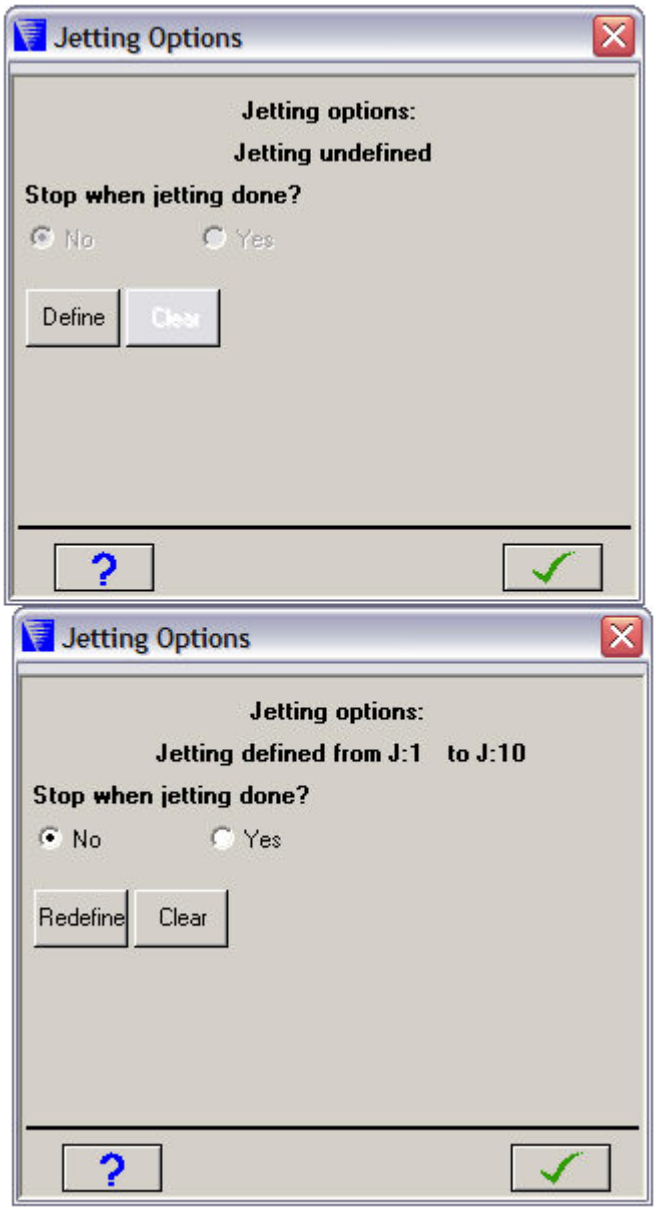
K-factor

The number of cells in the K direction you want to combine into a single cell. This total number of cells in the K direction (KMAX-1) must be exactly divisible by this number.

Dezone Range

In these six fields, enter the range over which the dezone should be performed.

10.13.10. Parts - Solvers (Structured) - Jetting



This window lets you set up jetting options for a 2D Shell Part.

These options let you perform shaped charge jetting analyses in which the liner is modeled as a shell Part and the jet characteristics are computed semi-empirically. The Jetting Tutorial describes these options in detail.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Jetting options

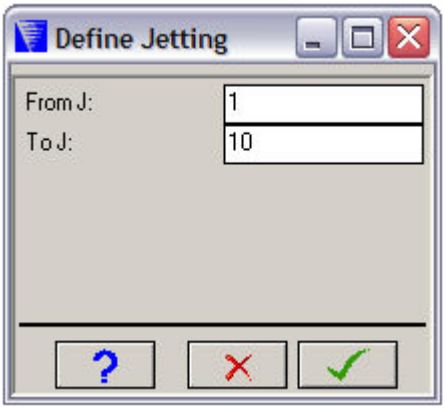
The top of this window shows the shell nodes you have defined to be jetting nodes.

Stop when jetting is done?

Select whether or not you want to stop the calculation once all the jetting nodes have jetted. If you select no, the calculation will continue until the wrap-up cycle or time you have set is reached.

Define / Redefine

Set the shell nodes you want to be jetting nodes. The following window will appear.



Enter the range of shell nodes you want to be jetting nodes.

Clear

Clear (undefine) all current jetting nodes.

10.14. Parts - Solvers (Unstructured)

This section discusses the following topics:

10.14.1. Parts - Solvers (Unstructured) - Solid
10.14.2. Parts - Solvers (Unstructured) - Shell
10.14.3. Parts - Solvers (Unstructured) - Beam

10.14.1. Parts - Solvers (Unstructured) - Solid

Clicking the **Solvers** button in the **Parts panel** displays the following dialog for unstructured Solid Parts in the lower region of the panel :

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

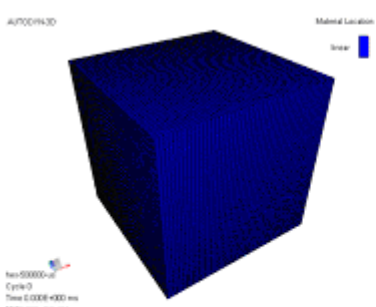
Hex Integration

The unstructured, 8 noded, constant strain, hexahedral element is available in two forms.

- Finite volume formulation with exact volume integration (Wilkins 1974) (Default)
- Finite element formulation using approximate Gaussian quadrature (Hallquist 1976)

The finite volume formulation is the default option and recommended for most applications involving large deformation or warped meshes. This formulation is the same as that used in the Structured (IJK) Lagrange solver.

The finite element formulation is included as a solver option and provides efficiency improvements over the above. However, the accuracy of the element is reduced for warped elements.



500,000 Elements

	Memory (Mbytes)	Time (Min)	Speed-up
IJK Hex	400	1.74	1.00
New Hex (Exact)	340	0.88	1.97
New Hex (Gauss)	340	0.62	2.80

3.2Ghz Pentium IV Processor, Windows XP

Example performance comparison for the three Hex solvers now available in Autodyn.

Hex Hourglass Control

Hourglass control for the Unstructured Hex solvers is provided in two forms:

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- AD standard
- Flanagan-Belytschko.

The "AD Standard" option is the default option and works well for most applications. This is the most efficient option in terms of memory and speed. A default viscous damping coefficient of 0.1 is recommended.

The "Flanagan-Belytschko" form of hourglass control is available as an option since the AD standard form may not perform well under large rigid body rotations. The Flanagan-Belytschko form of hourglass control is invariant under rotation hence overcomes this problem. Both viscous and stiffness based control is available. The default is stiffness based control with a coefficient of 0.1.

Tet Pressure Interaction

The Unstructured Tetrahedral element is available in three forms:

- Standard constant pressure (SCP) Tet
- Average nodal pressure (ANP) Tet, (Burton 1996).
- Nodal Based Strain (NBS) Tet, (Bonet *et al.*, 2001)

The SCP tetrahedral element is a basic, constant strain element and can be used with all the standard Autodyn material models including erosion. Explosive burn logic is also available. The element is intended as a "filler" element in meshes dominated by hexahedral elements. The element is known to exhibit locking behavior under both bending and constant volumetric straining (that is, plastic flow). If possible the element should therefore not be used in such cases.

The ANP tetrahedral element is an extension of the advanced tetrahedral element (Burton 1996) and can be used as a majority element in the mesh. The ANP tetrahedral overcomes problems of volumetric locking, which occur with the SCP tetrahedral element. However, the ANP tetrahedral element is still susceptible to shear locking in bending dominated problems. You should therefore be careful to verify their results in such cases.

For meshes containing a majority of tetrahedral elements, the ANP option is recommended.

Material Modeling Options Available with ANP-Tet Elements

Equations of State	Strength Models	Failure Models
Linear	Elastic	Hydro (Pmin)
Polynomial	Viscoelastic	Plastic Strain
Shock	Von Mises	Principal Stress
Porous	Johnson Cook	Principal Stress
Compaction	Piecewise JC	Principal Stress
P alpha	Zerilli Armstrong	Johnson-Holmquist
Rigid	Steinberg Guinan	RHT Concrete
Hyperelastic	Drucker-Prager	Grady Spall Model
	Johnson-Holmquist	Johnson Cook

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Equations of State	Strength Models	Failure Models
	RHT Concrete	Crack Softening
	MO Granular	
	Hyperelastic	

Parts containing ANP tetrahedral elements should only be filled with a single material. Multiple materials can be represented in a single body by creating multiple parts and joining them together.

The NBS tetrahedral element (Bonet *et al.*, 2001) is a further extension of the ANP tetrahedral element in that the entire solution is calculated on the nodes rather than just the pressure. The NBS tetrahedral element has the advantage over the other two tetrahedral formulations as it overcomes both volumetric and shear locking.

Supported material types in the NBS tetrahedral element are currently limited to ductile materials. The full list of supported material models can be found in the table below.

Table 10.1: Material Modeling Options Available with NBS-Tet Elements

Equations of State	Strength Models	Failure Models
Linear	Elastic	Hydro (Pmin)
Polynomial	Von Mises	Plastic Strain
Rigid	Johnson Cook	Principal Stress
Shock	Piecewise Johnson Cook	Principal Strain
	Zerilli Armstrong	Principal Stress/Principal Strain
	Cowper Symonds	Johnson-Holmquist
	Bilinear Hardening (Isotropic only)	Grady Spall
	Multilinear Hardening (Isotropic only)	Johnson Cook
	Steinberg Guinan	Stochastic Failure

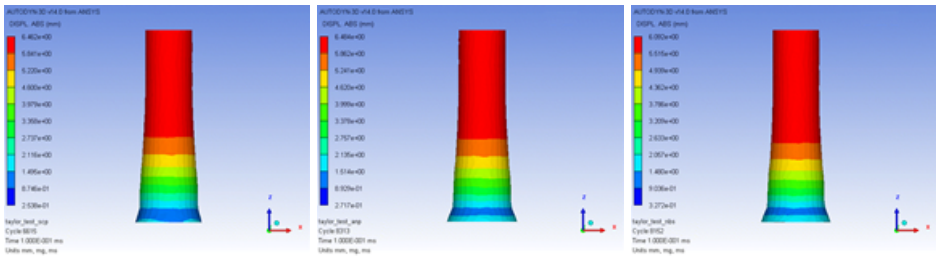
If a model containing NBS tetrahedral elements exhibits zero-energy modes (Puso, 2006), the PUSO stability coefficient can be set to a non-zero value. The recommended value is 0.1. Stabilization is achieved by taking a contribution to the nodal stresses from the SCP solution. Therefore, for models with a non-zero Puso stability coefficient, the solution is computed on both the nodes and the elements. Both can be visualized as contour plots: the standard variables show a volume-weighted nodal average in the elements, and the SCP solution used to stabilize for non-zero Puso coefficients can be plotted using the variables with suffix '.NBS'.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

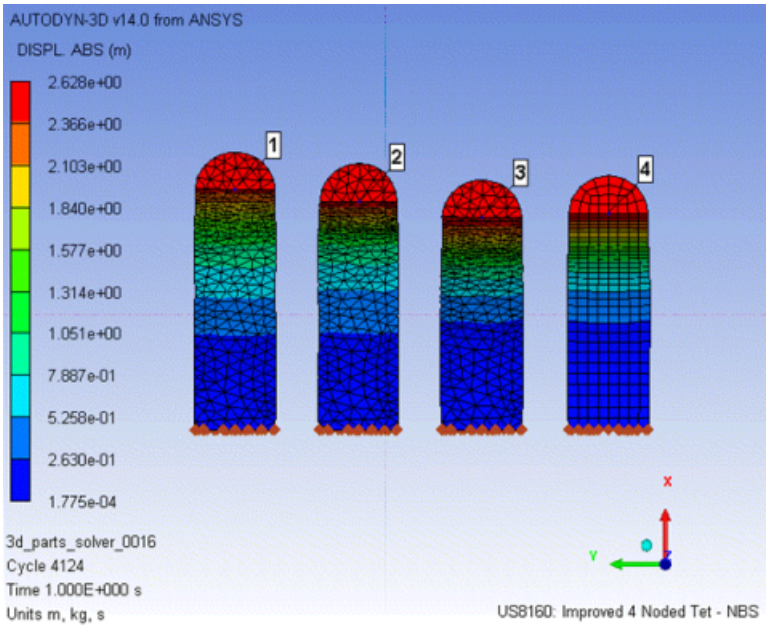
NBS tetrahedral elements cannot share nodes with ANP tetrahedral elements, SCP tetrahedral elements, shell elements or beam elements. Also note that the use of NBS tetrahedral elements with joins or spotwelds is not supported.

Note that a hexahedral mesh will generally provide more efficient results than a tetrahedral mesh hence we only recommend the use of predominantly tetrahedral mesh models for convenience of mesh generation.

Tet-SCP **Tet-ANP** **Tet-NBS**



Comparison of results of a Taylor test solved using SCP, ANP and NBS Tetrahedral elements. Results using NBS and ANP tetrahedral elements compare more favorably with experimental results.



Comparison of the performance of SCP (1), ANP (2), NBS tetrahedral (3), and hex (4) elements in a bending dominated problem. The displacement of the beam with NBS tetrahedral elements is the most similar to the beam meshed with hexahedral elements as it does not exhibit shear locking as is seen in the beams solved using SCP and ANP tetrahedral elements.

1. Burton A.J., "Explicit, Large Strain, Dynamic Finite Element Analysis with Applications to Human Body Impact Problems", PhD Thesis, University of Wales, December 1996.

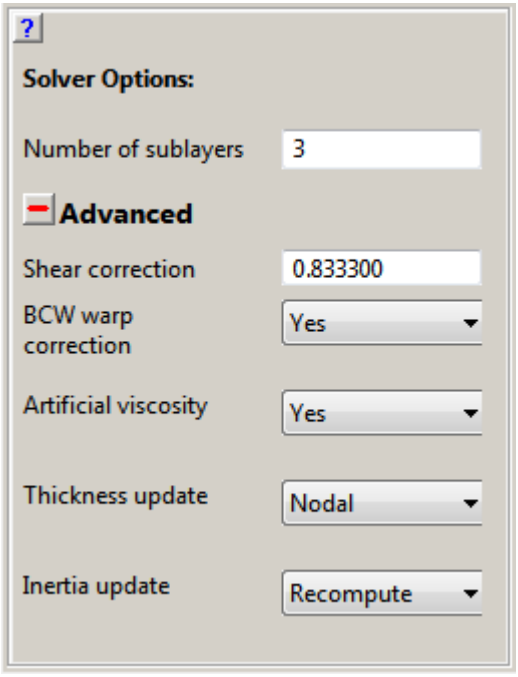
2. Bonet J, Burton A.J. "A simple averaged nodal pressure tetrahedral element for incompressible and nearly incompressible dynamic explicit applications". Communications in Numerical Methods in Engineering 1998; 14, 437-449.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

3. Bonet J., Marriott H., Hassan O. "An averaged nodal deformation gradient linear tetrahedral element for large strain explicit dynamics applications". *Communications in Numerical Methods in Engineering 2001*; **17**, 551-561.

4. Puso M. A.,Solberg J. "A stabilized nodally integrated tetrahedral". *International Journal for Numerical Methods in Engineering 2006*; **67**, 841-867.

10.14.2. Parts - Solvers (Unstructured) - Shell



Number of sublayers

Specify the number of through thickness integration points for an isotropic shell. The default of 3 is suitable for many applications however this number can be increased if better resolution of through thickness plastic deformation/flow is required.

The following advanced options are also available. Note these are not intended for general use.

Shear correction factor

The transverse shear in the element formulation is taken as constant over the thickness. The correction factor accounts for the replacement of the true parabolic variation through the thickness by a uniform transverse shear stress. We do not recommend that you change the default value.

BCW warp correction

The Belytschko-Lin-Tsay element formulation becomes inaccurate if the elements are warped. To overcome this shortcoming, the element formulation has a correction for the element warp included. We recommend that this correction is turned on.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Artificial Viscosity

Choose whether to apply artificial viscosity to compression waves in the shell. The values for the quadratic and linear coefficients and whether to use the linear term in the expansion come from the global settings for artificial viscosity which can be found under the **Damping Options** in the **Controls** menu.

Thickness Update

By default the thickness of a shell quad element is stored and updated at the nodes of the element. An alternative, slightly more efficient, option to store and update the thickness at the element is also provided. The thickness of a shell tri element is always updated at the element irrespective of the settings for Shell Thickness update in Autodyn.

Inertia Update

The principal axes of rotary inertia for shell elements are, by default, recalculated each cycle. Alternatively, there is an option to rotate the axes, rather than recompute each cycle. This latter option is more efficient, however can lead to instabilities.

10.14.3. Parts - Solvers (Unstructured) - Beam

Solution type

The response of a beam element to an applied load includes, by default, the effects of bending moments. This option lets you reduce the beam formulation to that of a truss, in which only axial and torsional effects are considered.

Timestep safety factor

Beam elements require a timestep safety factor that is less than the standard global factor defined under Controls, Timestep. A default value of 0.5 is used and should work for most applications. If problems are encountered with the stability of the beam elements, reducing this factor down to 0.1 is acceptable.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Volume element reinforcement

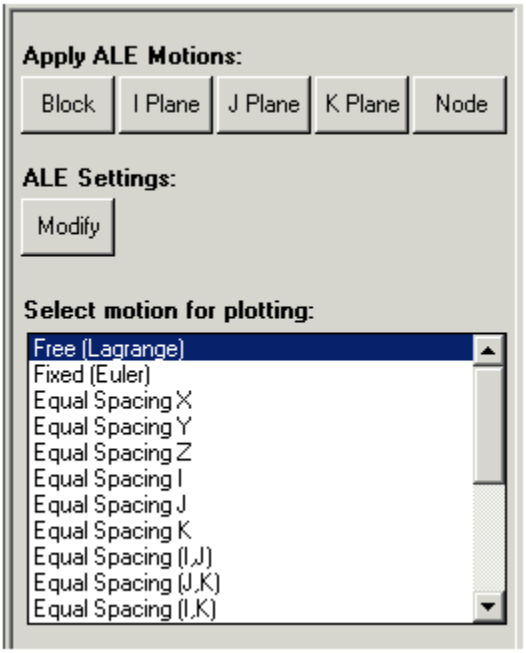
This option allows reinforcing bars, modeled with beam elements to be tied (bonded) to the volume of a solid element, without the restriction that the nodes of the beams and volume elements initially need to reside at the same physical location. The bonded beam nodes are constrained to stay at the same initial parametric location within the volume element during element deformation. Typical applications involve reinforced concrete or reinforced rubber structures like tires and hoses.

At start up, Autodyn will determine the parametric positions of all nodes of beam Parts, selected as reinforcement (Parts, Solver), within volume elements (hex, tet, penta) in the model. The nodes are tied to the parametric co-ordinate of the volume element until the element erodes.

On erosion of the volume element, the beam node bonding constraint is removed and becomes a free beam node.

10.15. Parts - Motions

Clicking the **Motions** button in the **Parts panel** displays the following dialog for structured Parts in the lower region of the panel :



This dialog lets you apply motion constraints to an **ALE** Part.

Block

Apply motion constraints to an IJK block of nodes.

I/J/K Plane

Apply motion constraints to an I, J, or K plane of nodes.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Node

Apply a motion constraint to a single node.

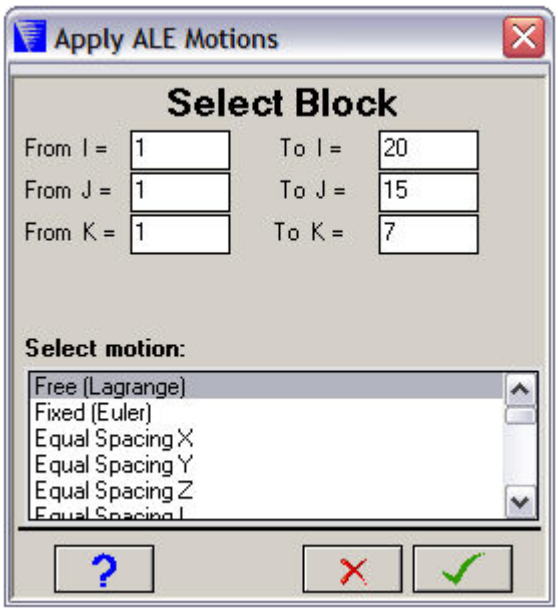
Modify (ALE Settings)

Change various ALE settings.

Select motion for plotting

Select the motion constraint you want to plot in this scrolling box. All nodes with the selected motion constraint will be highlighted in the plot.

10.15.1. Parts - Motions - Block



This window lets you apply an ALE motion constraint to a block of nodes in IJK space. The example shown is for applying a motion constraint to an I block (input for the other blocks is similar).

Select Block

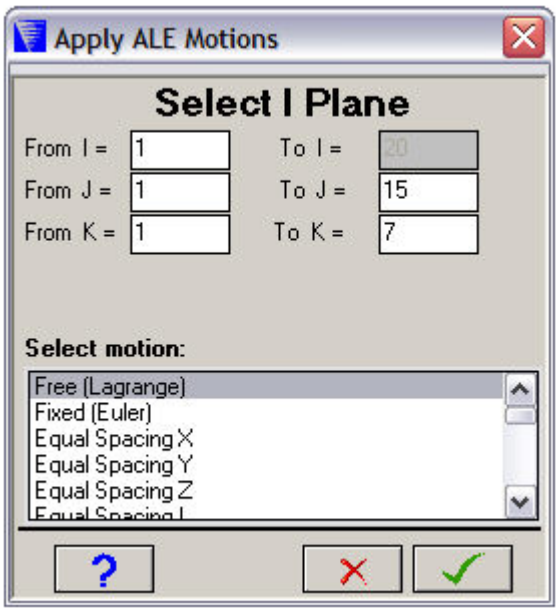
The I index for the block and the (J,K) range for the nodes to which you want to apply the motion constraint.

Select motion

The motion constraint you want to apply.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

10.15.2. Parts - Motions - I / J / K Plane



This window lets you apply an ALE motion constraint to a plane of nodes in IJK space. The example shown is for applying a motion constraint to an I plane (input for the other planes is similar).

Select Plane

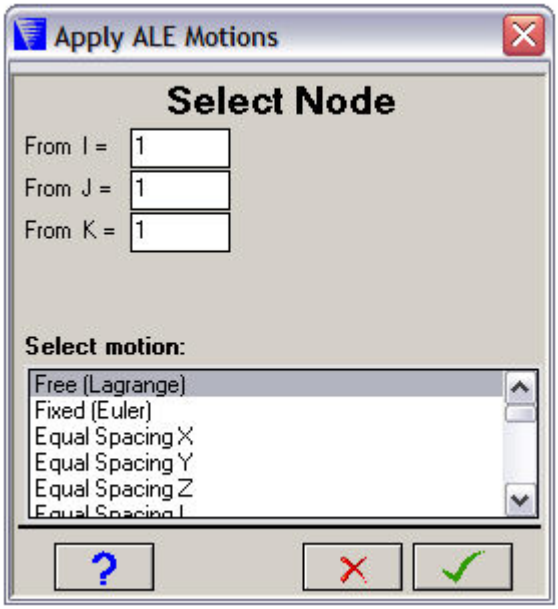
The I index for the plane and the (J,K) range for the nodes to which you want to apply the motion constraint.

Select motion

The motion constraint you want to apply.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

10.15.3. Parts - Motions - Node



This window lets you apply an ALE motion constraint to a node in IJK space.

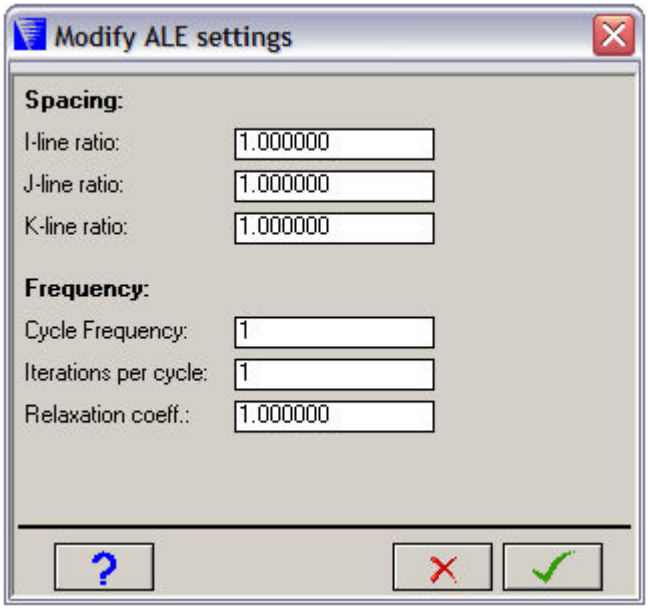
Select Node

The (I,J,K) indices for the node to which you want to apply the motion constraint.

Select motion

The motion constraint you want to apply.

10.15.4. Parts - Motions - Modify



Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

This window lets you change ALE settings.

Spacing

The geometric ratios for spacing of I, J, and K lines. These are used only for certain motion constraints.

Frequency

Cycle Frequency

Enter how often (in cycles) you want ALE rezoning to be performed.

Iterations per cycle

Enter the number of iterations you want the ALE solver to perform in converging to the spacing you have requested. This is used for all motion constraints except **Fixed (Euler)**.

Relaxation coefficient

Enter the relaxation coefficient you want to use for the ALE rezoning.

A value of 1.0 will move each node exactly to the location required for its motion constraint.

A value of 0.5 will move each node 50% of the distance towards the location required for its motion constraint.

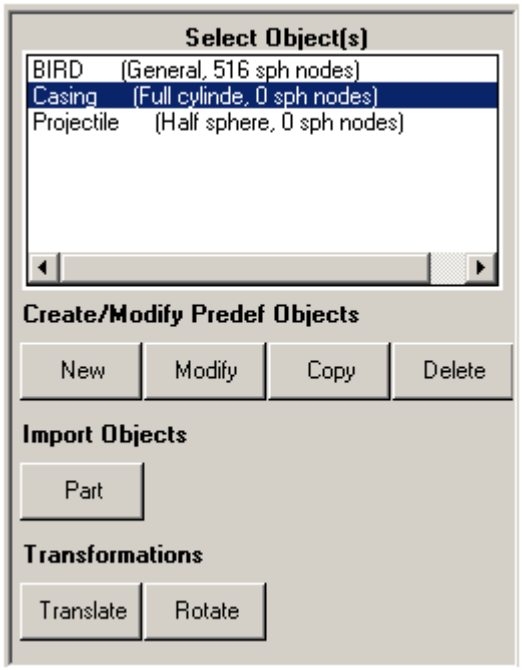
10.16. Parts - Geometry

This section discusses the following topics:

10.16.1. Parts - Geometry - SPH
10.16.2. Parts - Geometry - New/Modify
10.16.3. Parts - Geometry - Copy
10.16.4. Parts - Geometry - Import Part
10.16.5. Parts - Geometry - Translate
10.16.6. Parts - Geometry - Rotate

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

10.16.1. Parts - Geometry - SPH



This panel allows you to define the geometry for SPH objects.

Select Object(s)

All defined SPH objects are shown in the Select Object(s) list. Select a single object to Modify, Copy or Delete, or select one or more objects to Transform.

Create/Modify Predef Objects

New

Define a new SPH object.

Modify

Modify an existing SPH object. This button is active only if a single SPH object is selected. You cannot modify objects created by conversion from a part (using the import objects - part button).

Copy

Create a copy of the selected SPH object. This button is active only is a single SPH object is selected.

Delete

Delete the selected SPH object. This button is active only if a single SPH object is selected.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Import Objects

Part

Covert an existing part (structured or unstructured) into an SPH object. This operation is useful for creating complex SPH objects.

Transformations

Translate

Translate the currently selected objects.

Rotate

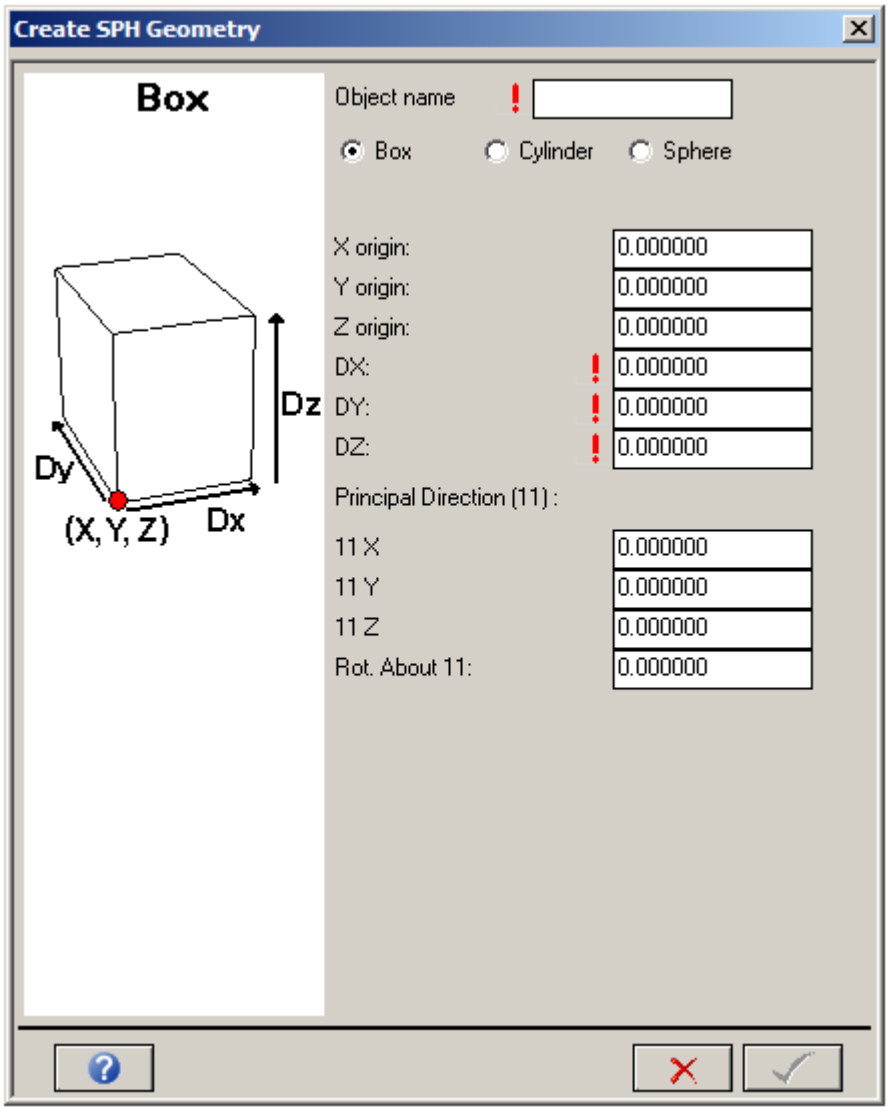
Rotate the currently selected objects.

10.16.2. Parts - Geometry - New/Modify

There are three types of Geometry Predef objects you can use - Box, Cylinder and Sphere. These are described below.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

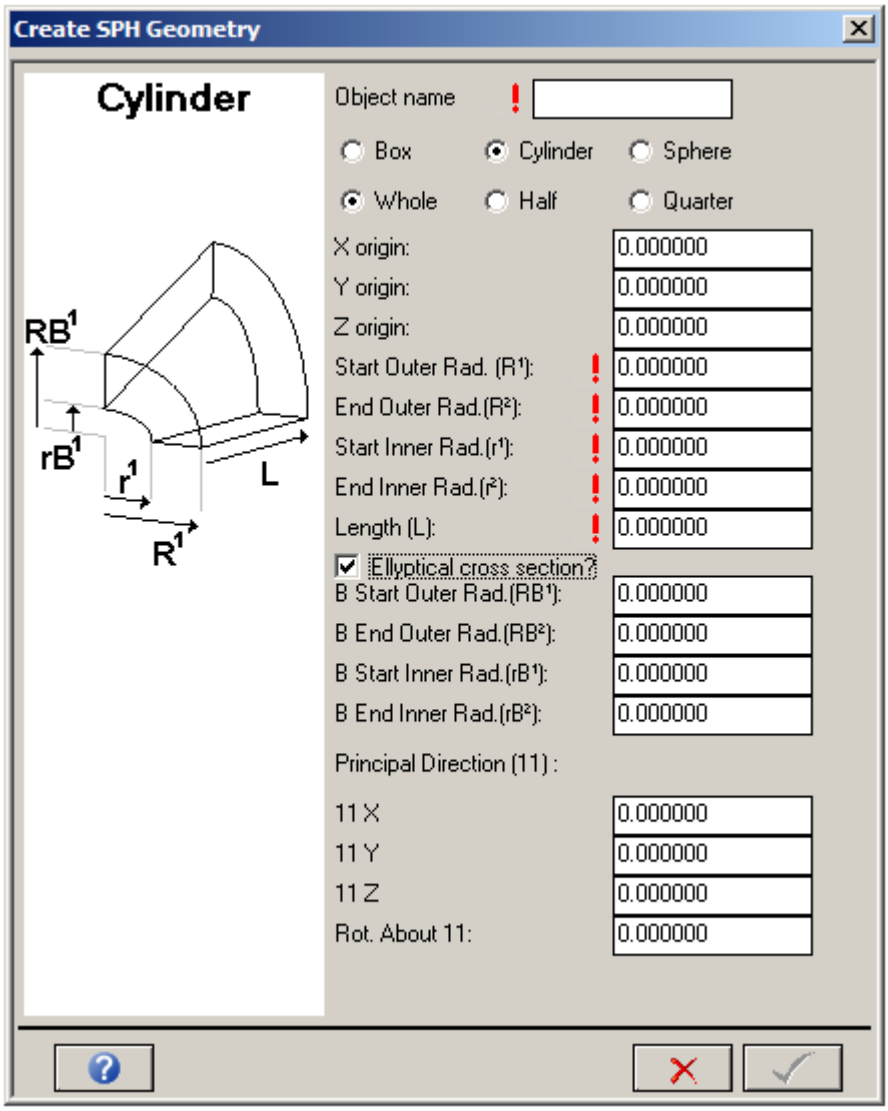
Box



Define the box by entering the XYZ coordinates of the origin, and the size in each of its three directions. The orientation of the box is defined by a principal direction vector and a rotation about this vector. When the box is defined, a wire-frame representation is shown on the screen, together with an arrow displaying the principal vector direction.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Cylinder



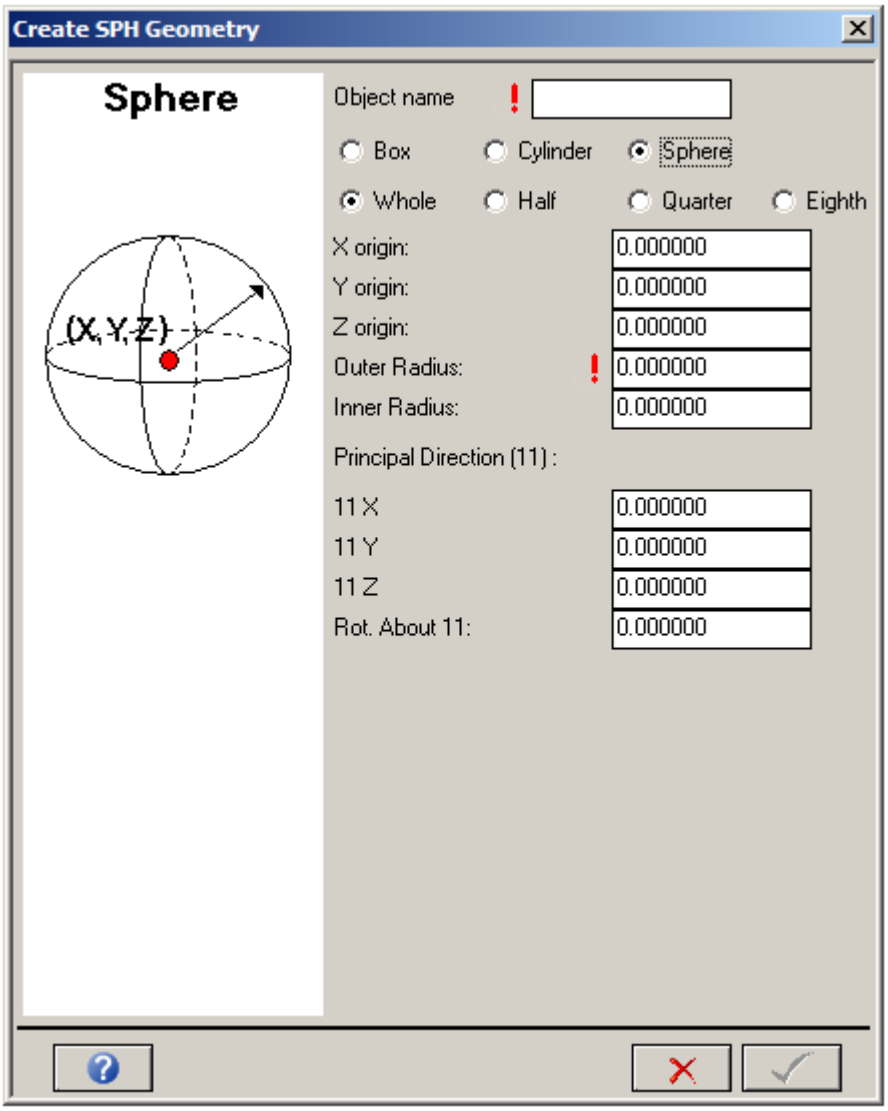
You may define either a whole, half or quarter cylinder by selecting the appropriate option. To define the cylinder, first enter the XYZ coordinates of the start of the cylinder axis. You are then prompted for the start and end radius of the cylinder. If you want to define a hollow cylinder, enter inner radii for the start and end of the cylinder; to define a solid cylinder, leave these two values as zero.

You may also define a cylinder that is elliptical in cross-section. To do this you must provide extra radii information. This is indicated on the figure on the left hand side of the window.

The orientation of the cylinder is defined by a principal direction vector, which corresponds to the axis of the cylinder and a rotation about this vector. When the cylinder is defined, a wireframe representation is shown on the screen, together with an arrow displaying the principal vector direction.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Sphere

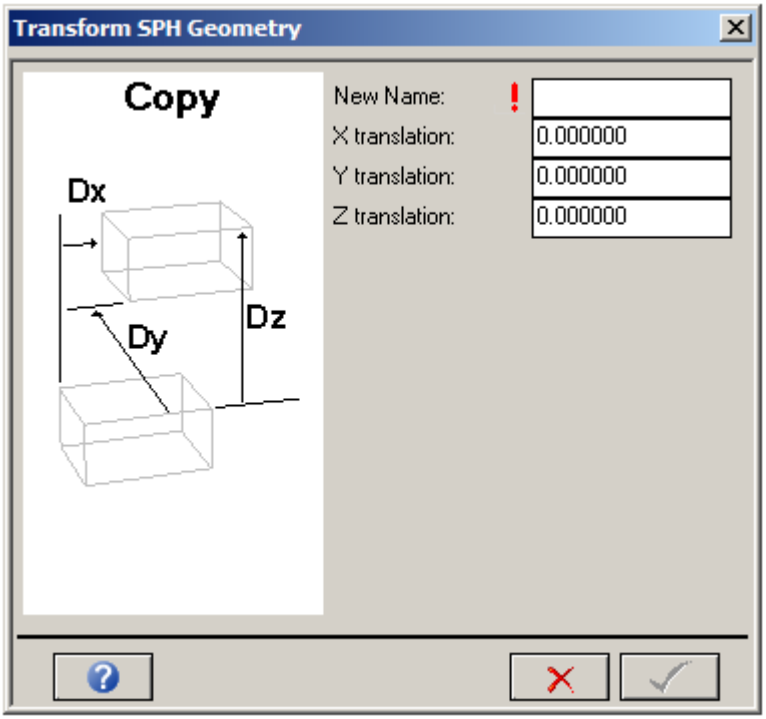


You may define either a whole, half, quarter or eighth sphere by selecting the appropriate option. To define the sphere, enter XYZ coordinates of the sphere origin and an outer radius. You may specify a hollow sphere by entering an inner radius.

The orientation of the sphere is defined by a principal direction vector and a rotation about this vector. When the sphere is defined, a wireframe representation is shown on the screen, together with an arrow displaying the principal vector direction.

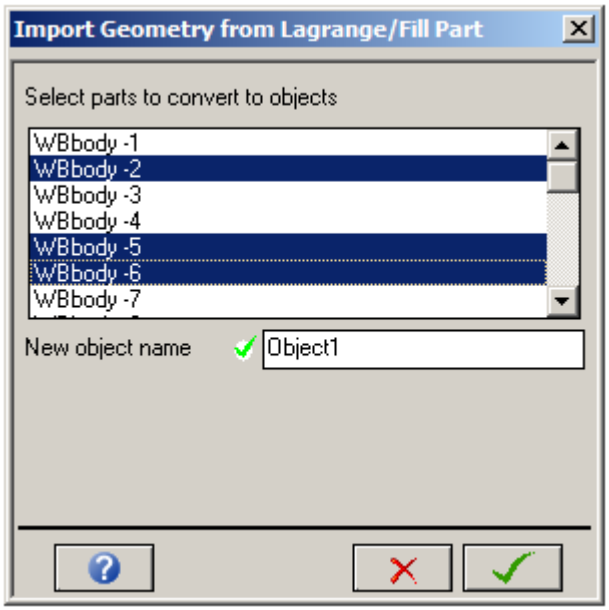
Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

10.16.3. Parts - Geometry - Copy



This window lets you create a copy of the currently selected SPH object. Enter the name of the new SPH object, and also an offset in the X, Y, and Z direction.

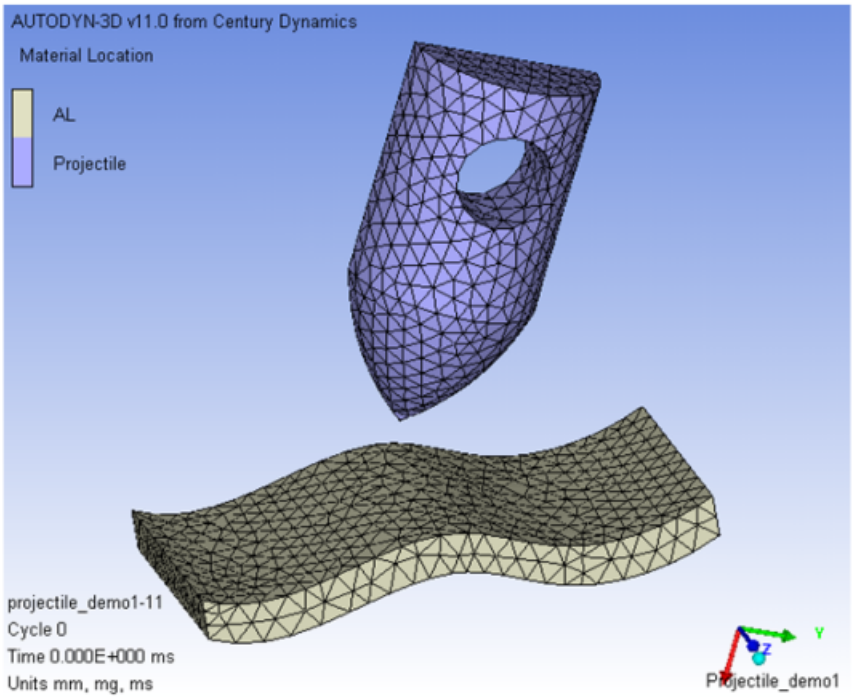
10.16.4. Parts - Geometry - Import Part

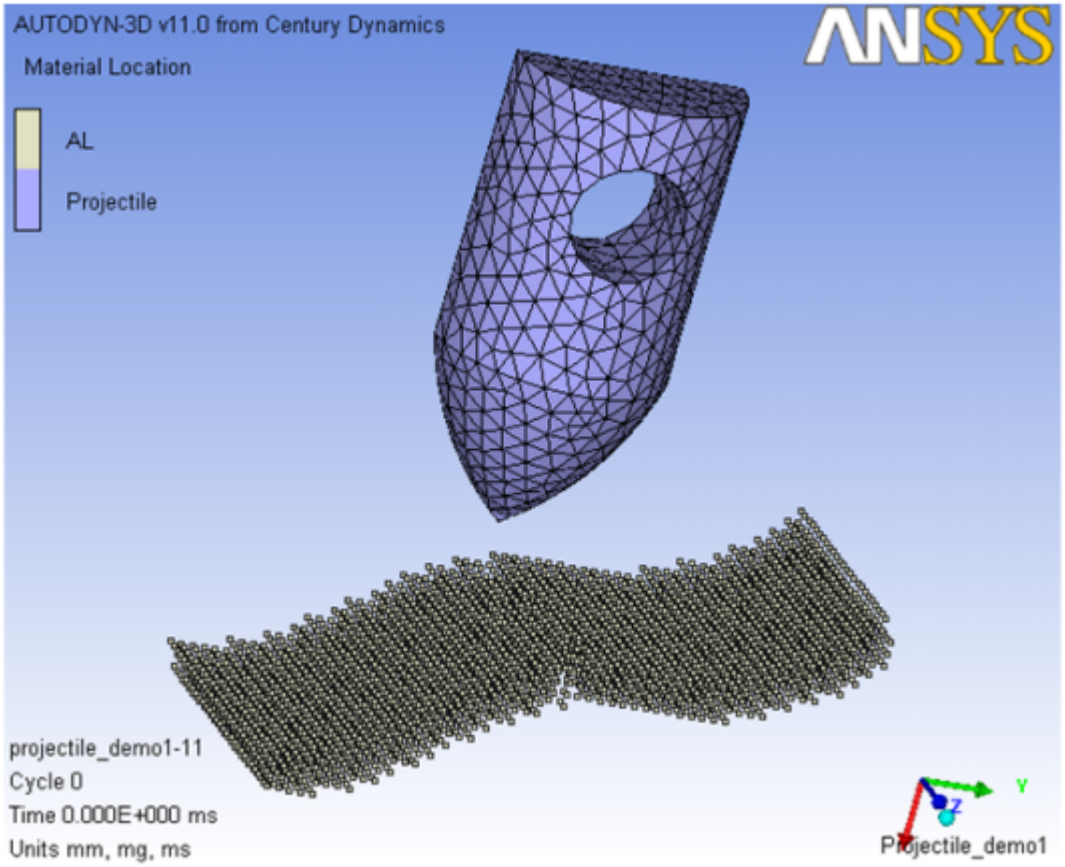


The parts list contains all structured Lagrange or unstructured volume parts in your model. Select one or more of these parts to create an object that can then be packed with SPH particles. You must enter a name for the new object to be created.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

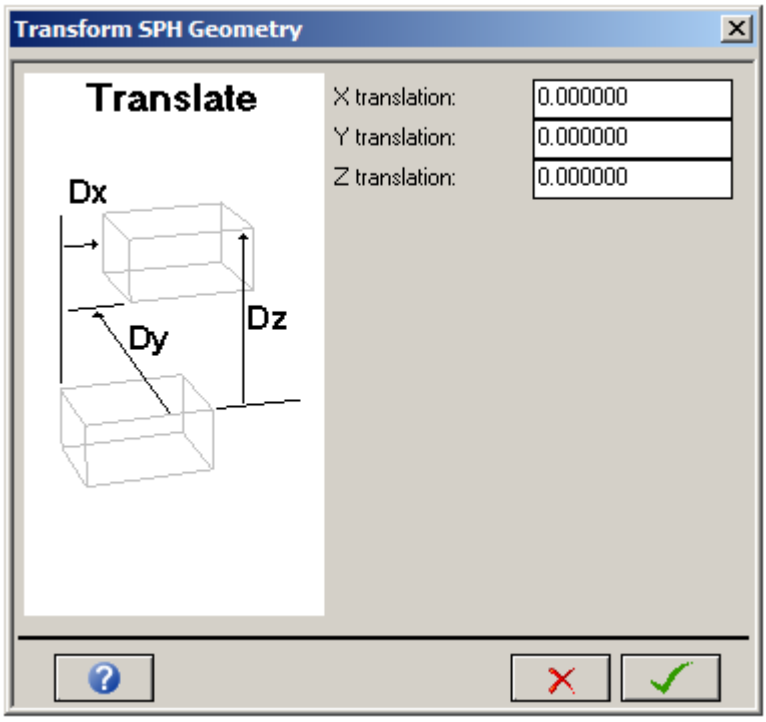
The images below show an example of a model with a projectile impacting a target. The target modeled with tetrahedral volume elements in the first image has been converted to an SPH object and packed with SPH nodes. This is shown in the second image.





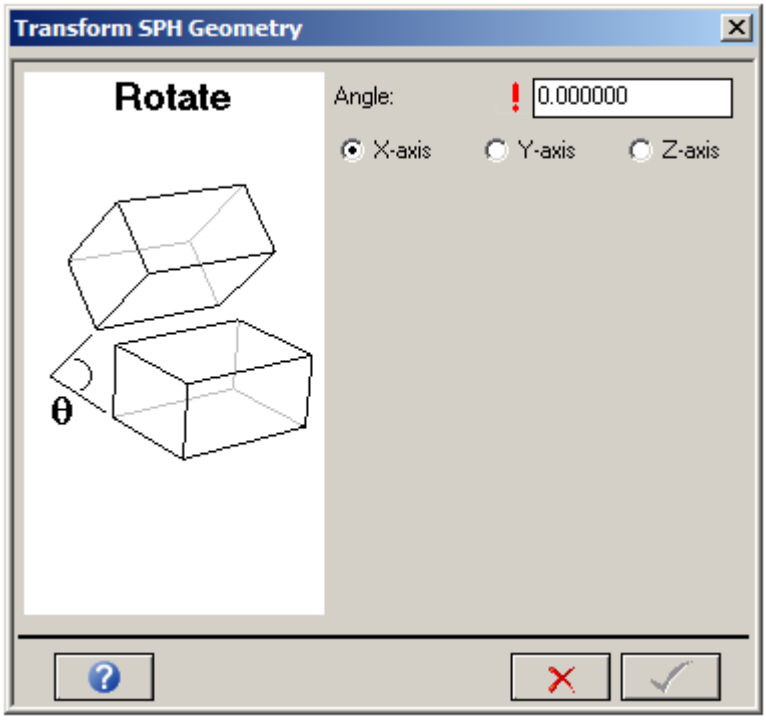
Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

10.16.5. Parts - Geometry - Translate



Enter the distance in the X, Y, and Z direction to translate the currently selected part or parts.

10.16.6. Parts - Geometry - Rotate



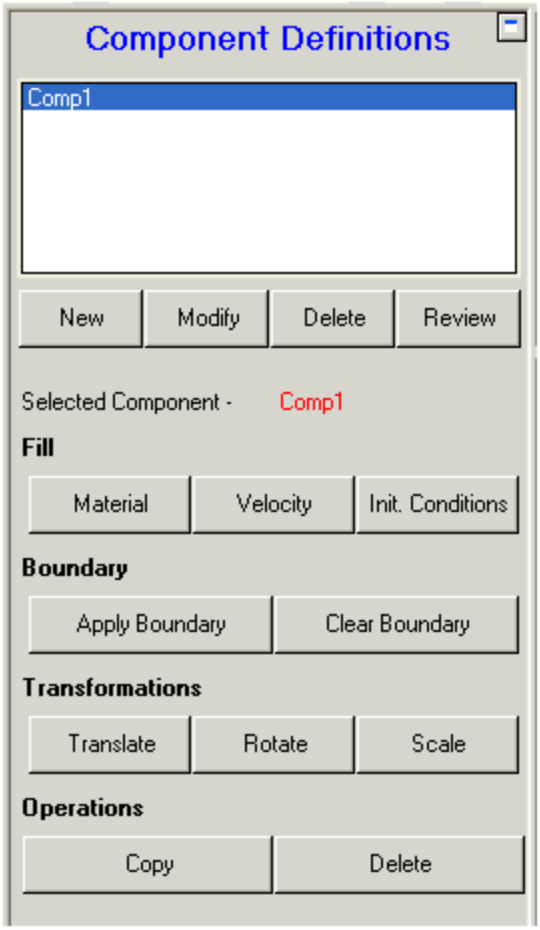
Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Specify the angle (in degrees) defining the rotation about the selected axis (either X, Y or Z).

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information
of ANSYS, Inc. and its subsidiaries and affiliates.

Chapter 11: Autodyn Components



A Component is a group of structured and unstructured Parts to which you can collectively apply operations.

This panel lets you define Components and use them to perform various operations.

Component List

- The top box displays a list of the Components you have defined. You can select a Component in this list.

  New creates a new Component.

- **Modify** enables editing of an existing Component.
- **Delete** removes a Component.
- **Review** enables review existing Components.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Fill

- **Material** fills all Parts of the selected Component with a material.
- **Velocity** assigns a velocity to all Parts of the selected Component.
- **Initial Conditions** assigns initial conditions to all Parts of the selected Component.

Boundary

- **Apply Boundary** applies a boundary condition to the selected Component.
- **Clear Boundary** clears a boundary condition from the selected Component with a material.

Transformations

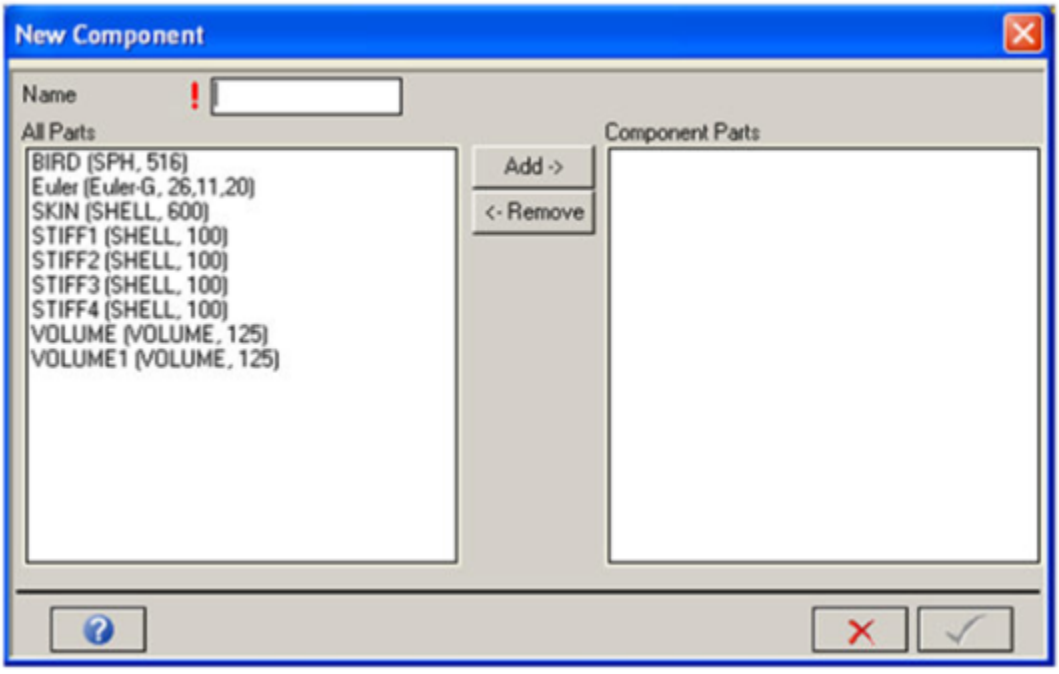
- **Translate** all Parts of the selected Component.
- **Rotate** all Parts of the selected Component.
- **Scale** all Parts of the selected Component.

Operations

- **Copy** the selected Component.

  Note that unstructured Parts cannot be copied.
- **Delete** all Parts of the selected Component.

11.1. Components - New



Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

This window lets you create a new Component.

Name

The name of your Component.

All Parts

Initially shows a list of all the Parts in your model. You can select one or more Parts in this list.

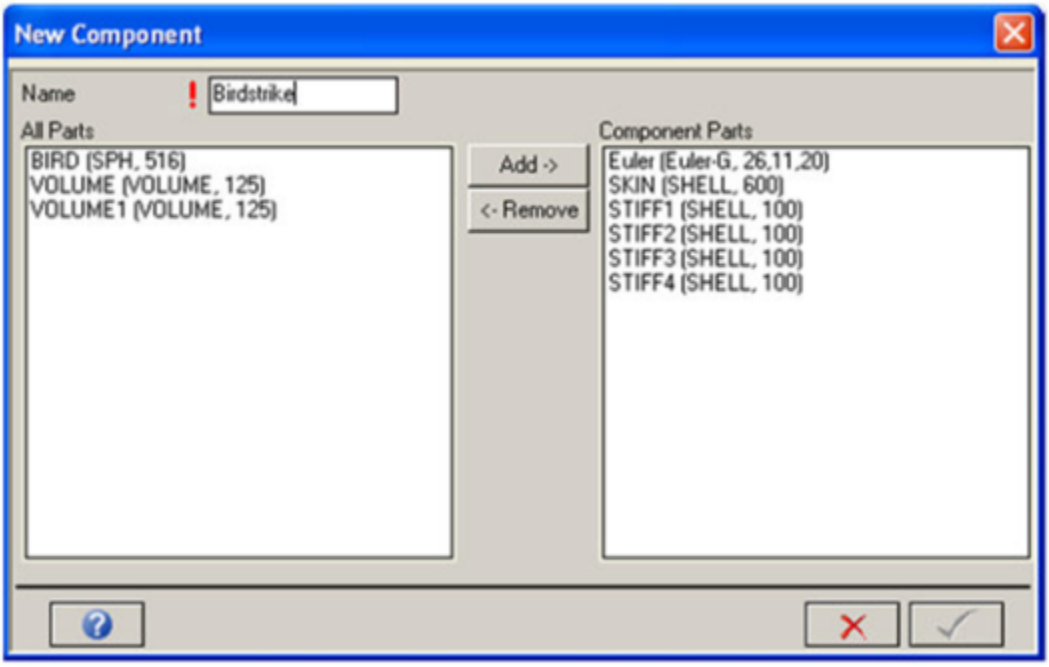
Component Parts

A list of the Parts you have chosen for your Component.

Add

Select one or more Parts in the left box and click this button to add the selected Parts to your Component.

The Parts will be removed from the list in the left box and added to the list in the right box, as shown in the following example:

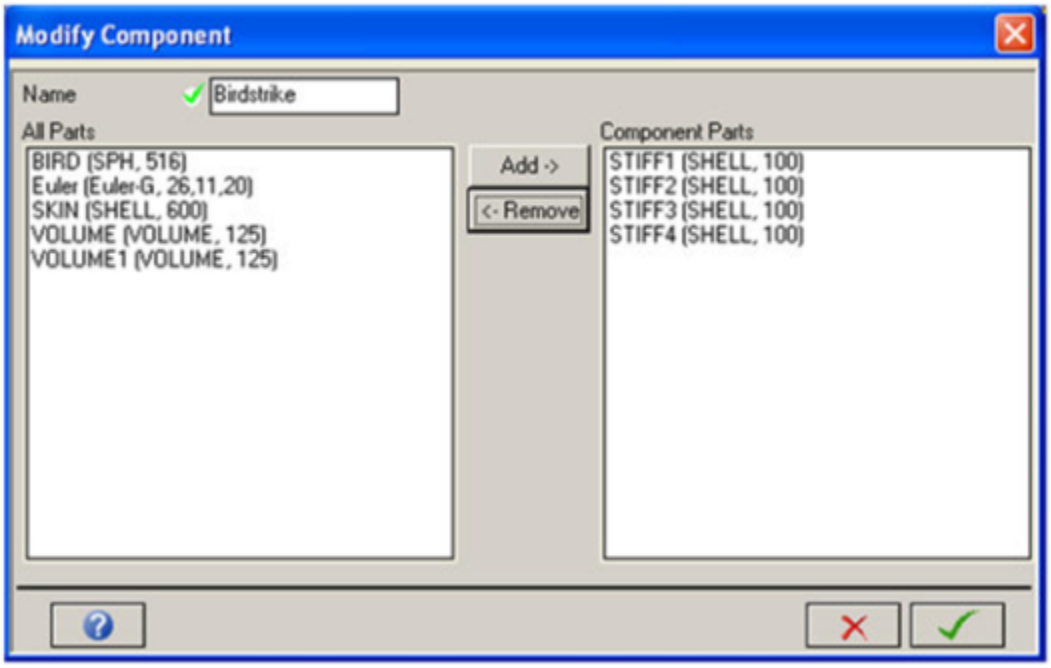


Remove

Select one or more Parts in the right box and click this button to remove the selected Parts to your Component.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

11.2. Components - Modify



This window lets you modify a Component.

Name

Change the name of your Component.

All Parts

List of all the Parts in your model not currently in your Component. You can select one or more Parts in this list.

Component Parts

List of the Parts you have chosen for your Component. You can select one or more Parts in this list.

Add

Select one or more Parts in the left box and click this button to add the selected Parts to your Component. The Parts will be removed from the list in the left box and added to the list in the right box.

Remove

Select one or more Parts in the right box and click this button to remove the selected Parts to your Component. The Parts will be removed from the list in the right box and added to the list in the left box.

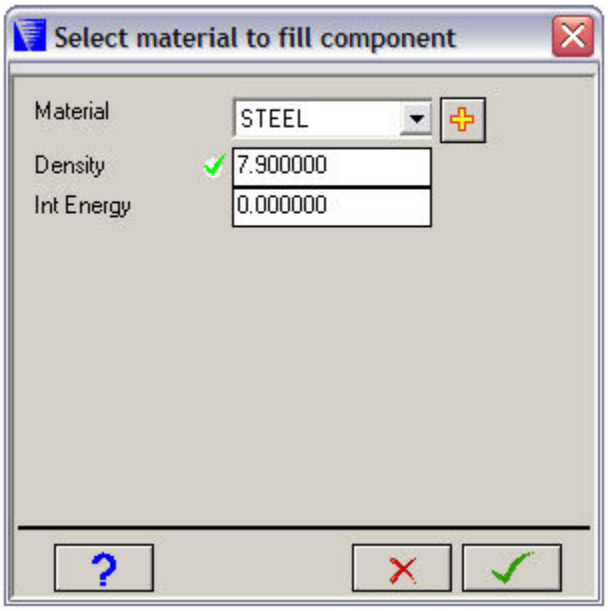
Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

11.3. Components - Delete

This window lets you delete one or more components.

Select the components you want to delete in the list and click .

11.4. Components - Material



Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

This window lets you fill all Parts of the selected Component with a material.

Material

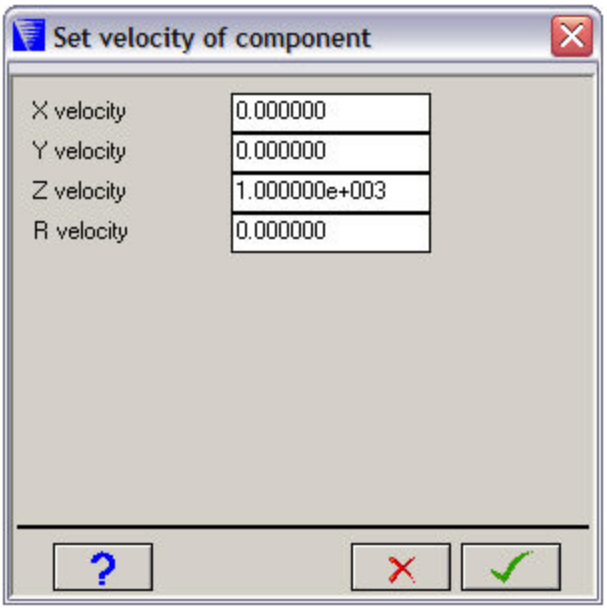
The material you want to use to fill your Component.

You can define new materials by clicking the Materials button on the Navigation Bar or by clicking next to the option in this window.

Material Parameters

The **Density** and **Energy** values for your fill.

11.5. Components - Velocity

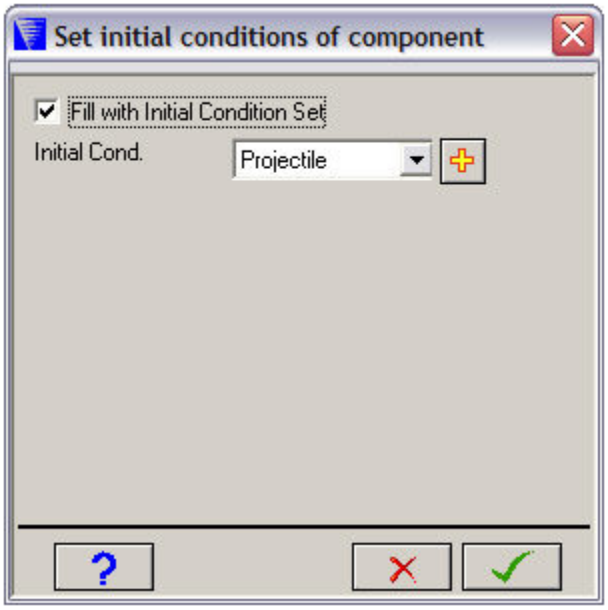


This window lets you assign a velocity to all Parts of the selected Component.

Enter the velocity components in the fields provided and click .

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

11.6. Components - Initial Conditions



This window lets you assign initial conditions from an Initial Condition Set to all Parts of the selected Component.

Fill with Initial Condition Set

Check this box to fill the Component using an Initial Condition Set.

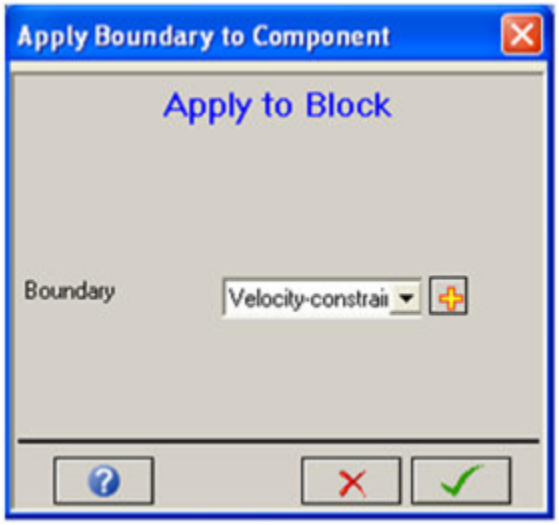
The advantage of filling Parts using an Initial Condition Set is that if you later change one or more parameters in the set, you do not have to refill your Part.

Use the pull-down menu to select the Initial Condition Set you want to use.

You can define new Initial Condition Sets by clicking the **Init.Cond.** button on the Navigation Bar or by clicking next to the option in this window.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

11.7. Components - Apply Boundary



This window lets you apply a boundary condition to a Component.

Boundary

Select the boundary condition you want to apply from this pull-down menu.

You can define Boundary Conditions by clicking the **Boundaries** button in the Navigation bar, or you can click next to the pull-down menu.

11.8. Components - Clear Boundary

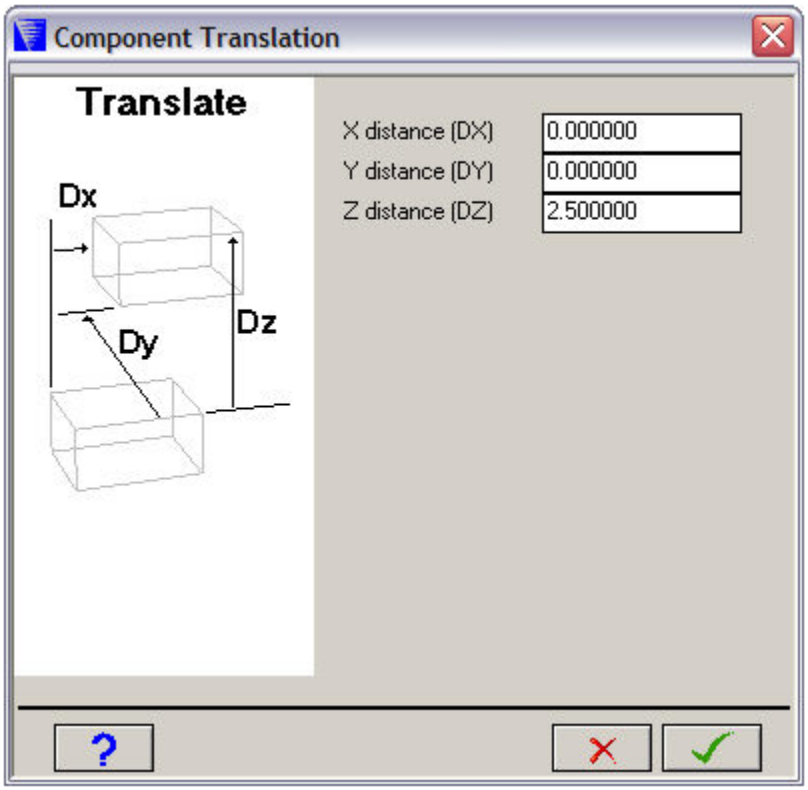
This window lets you clear a Boundary Condition from the current Component.

This operation does not delete the definition of the Boundary Condition, so you can re-apply it later.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Select the boundary condition you want to clear from the pull-down menu and click .

11.9. Components - Translate



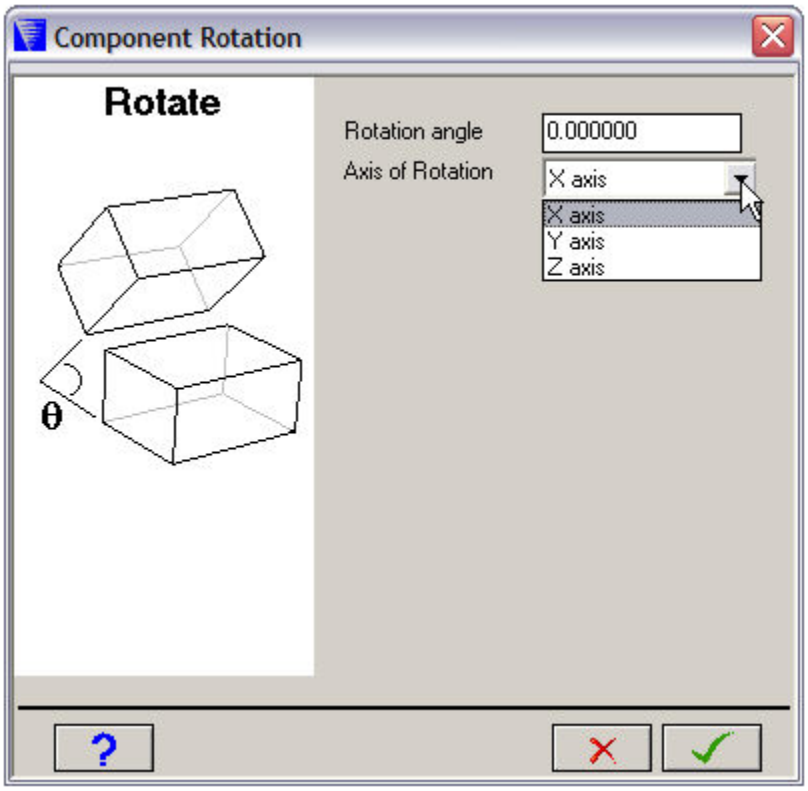
This window lets you translate all Parts of the selected Component in XYZ space.

X/Y/Z distance

The X, Y and Z components of the translation.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

11.10. Components - Rotate



This window lets you rotate all Parts of the selected Component about an axis in XYZ space.

Rotation

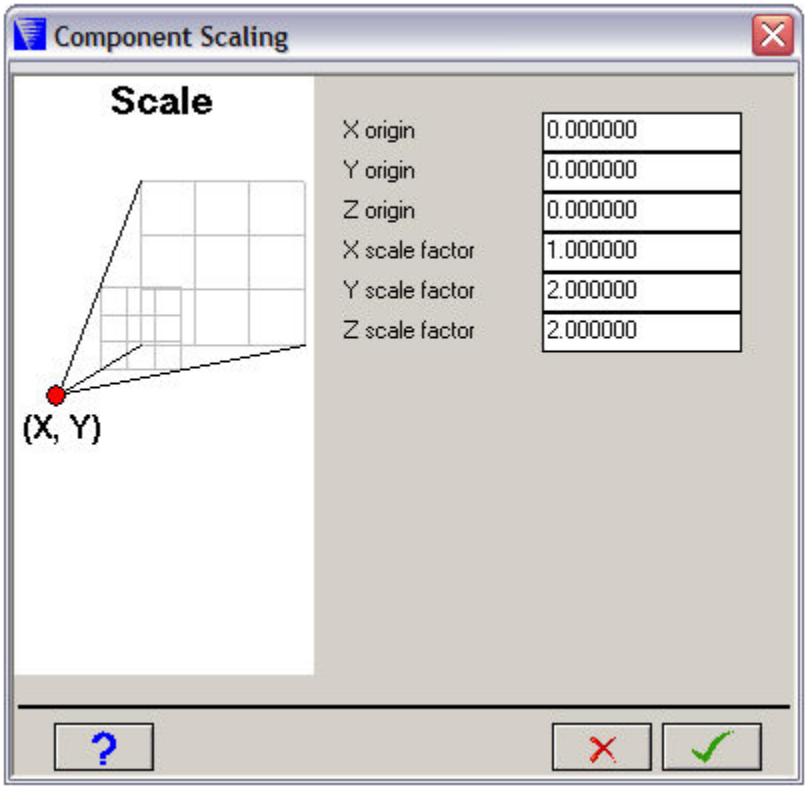
The rotation angle in degrees.

X/Y/Z-axis

The axis about which the Component will be rotated.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

11.11. Components - Scale



This window lets you scale all Parts of the selected Component in XYZ space.

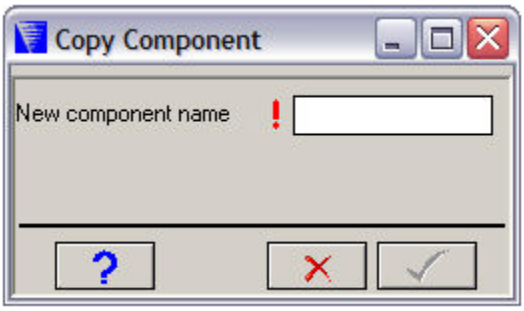
Origin

Input the X, Y, and Z coordinates of the origin for the scaling.

Scale Factors

Input the X, Y, and Z scale factors.

11.12. Components - Copy



This window lets you copy the selected Component to a new Component.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

During the copy operation a new copy of each structured part in the Component is generated.

Unstructured Parts cannot be copied and therefore will not be present in the copy of the Component.

Enter your name for the new Component and click .

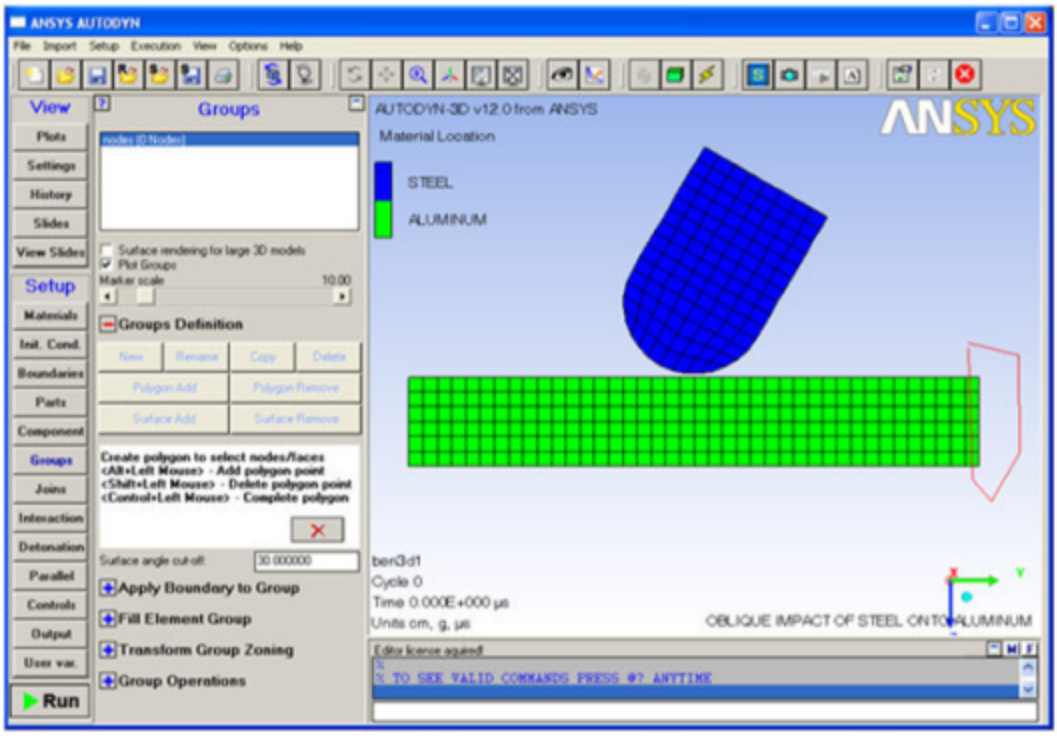
Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Chapter 12: Autodyn Groups

Introduction

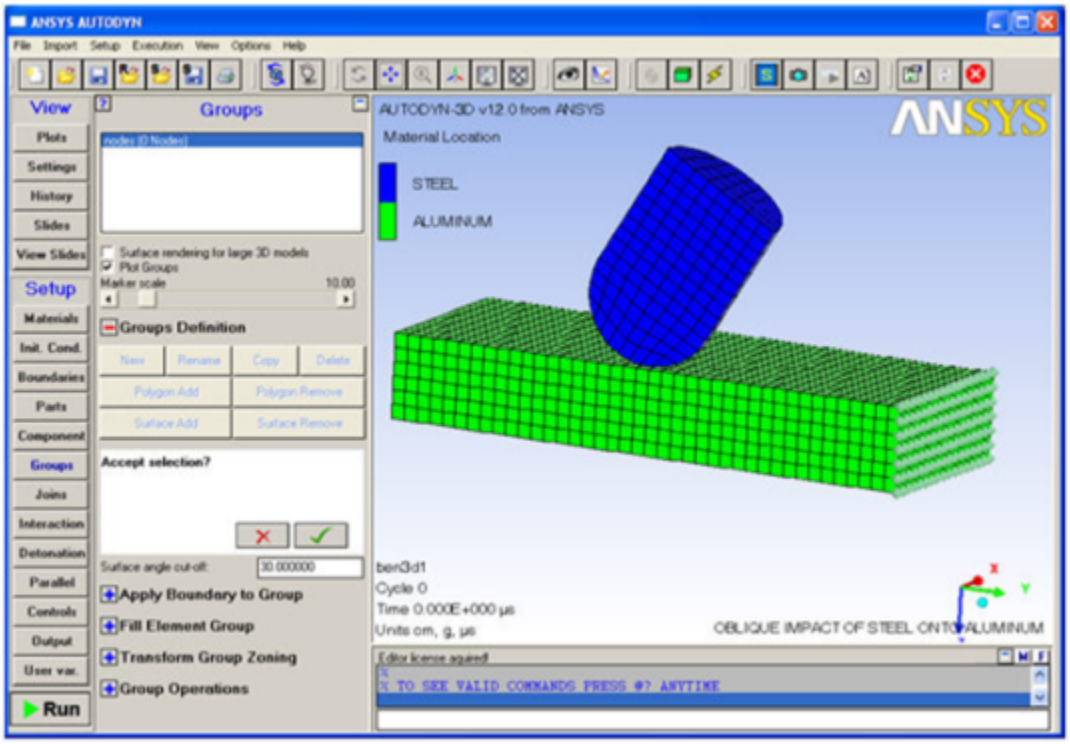
The Group option enables you to group an arbitrary (independent of Part and Component) selection of nodes faces or elements on which operations can subsequently be performed.

An interactive method of selecting groups of nodes, faces or elements is available. The selection is made by drawing a 2D polygon on the screen around the region of interest, as shown in the figure below.



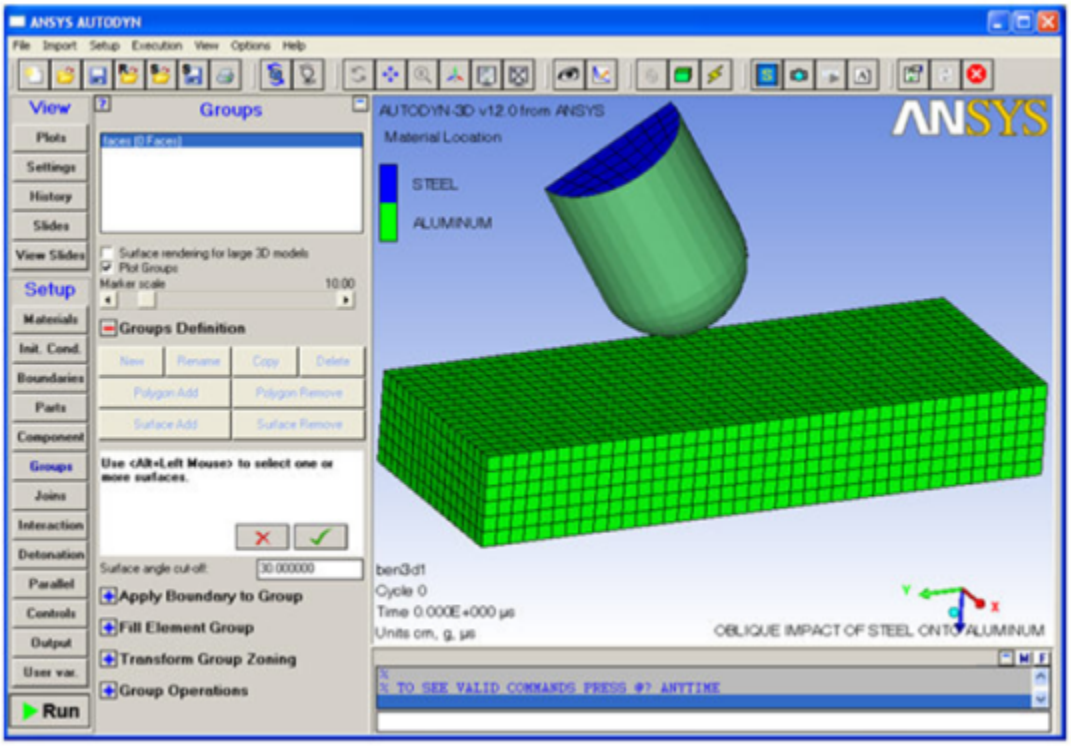
Any nodes, faces or elements that are enclosed by the polygon are highlighted, and you are prompted to accept the selection of nodes or faces. These are then added to the current group.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



For face groups, an alternative surface selection method exists. The selection is made by interactively picking a face element in the graphics window. Any faces that are joined to the selected face and have a normal angle between the two faces of less than a user specified value are then added to the group.

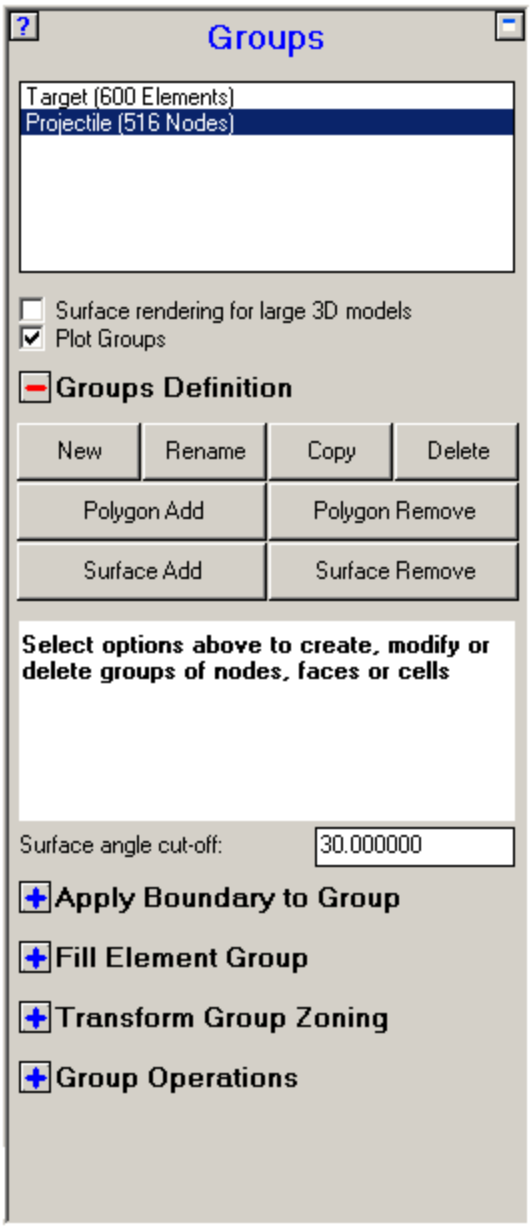
For example, selecting an element on the side of a cube would add all faces on that side of the cube. By setting the surface angle cut-off to a sufficiently large value, curved surfaces may also be selected, as shown in the example below, where the curved surface of the projectile has been added to a group.



Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Once a group is defined, a number of operations can be performed, depending on the type of group.

Groups



A Group is a selection of nodes, faces or elements to which you can perform operations such as applying boundaries and filling.

This panel lets you define Groups and use them to perform various operations.

Group List

Displays a list of the Groups you have defined, together with their type (node, face or element) and the group size. You can select a Group in this list.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

New

Create a new Group.

Rename

Rename an existing Group.

Delete

Delete a Group

Review

Review existing Groups.

Groups Definition

Polygon Add

Click this button to interactively define a polygon; any nodes, faces or elements that lie within the 2D polygon are added to the selected Group.

Select the polygon points using the **Alt**+left mouse button. To delete the last polygon point, use **Shift**+left mouse button. To complete the polygon, use **Ctrl**+left mouse button.

After completing the polygon, the selected nodes or faces are displayed. Click the button to accept this selection, and add to the current Group. At any stage, the button may be used to cancel the process.

To be able to use the Polygon Add option you should use the zoom option, available from the toolbar of Autodyn.

Note:

The Polygon Add option is not available (greyed-out) when you have zoomed into the model using the rubber-band option using **Ctrl**+left mouse button. The Polygon Add option is also not available when the Shell Thickness plot is active.

Polygon Remove

Click this button to interactively define a polygon; any nodes, faces or elements that lie within the 2D polygon are removed from the selected Group.

Select the polygon points using **Alt**+left mouse button. To delete the last polygon point, use **Shift**+left mouse button. To complete the polygon, use **Ctrl**+left mouse button.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

After completing the polygon, the selected nodes or faces are displayed. Click the button to accept this selection, and add to the current Group. At any stage, the button may be used to cancel the process.

To be able to use the Polygon Remove option you should use the zoom option, available from the toolbar of Autodyn.

Note:

The Polygon Remove option is not available (greyed-out) when you have zoomed into the model using the rubber-band option using **Ctrl**+left mouse button. The Polygon Remove option is also not available when the Shell Thickness plot is active.

Surface Add

Click this button to choose a surface and add its nodes, faces or elements to the selected Group.

Select a face using Alt and the left mouse button. Any faces that are joined to the selected face and have a normal angle between the two faces of less than a value you specify are then added to the group. For example, selecting an element on the side of a cube would add all faces on that side of the cube. By setting the surface angle cut-off to a sufficiently large value, curved surfaces may also be selected.

After selecting a face, the corresponding surface is highlighted in the graphics window. Click the button to accept this selection or continue adding further surfaces with the **Alt**+left mouse button. At any stage, the button may be used to cancel the process.

Surface Remove

Click this button to choose a surface and remove its nodes, faces or elements to the selected Group.

Select a face using Alt and the left mouse button. Any faces that are joined to the selected face and have a normal angle between the two faces of less than a value you specify are then removed from the group. For example, selecting an element on the side of a cube would remove all faces on that side of the cube. By setting the surface angle cut-off to a sufficiently large value, curved surfaces may also be selected.

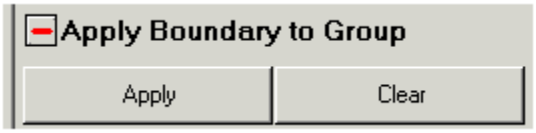
After selecting a face, the corresponding surface is highlighted in the graphics window. Click the button to accept this selection or continue adding further surfaces with **Alt**+left mouse button. At any stage, the button may be used to cancel the process.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Surface Angle Cut-Off

Enter a cut-off angle (in degrees). Only faces with an angle between the normal and a neighbor face's normal of less than this value will be included in a surface selection.

Apply Boundary to Group



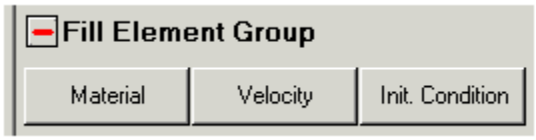
Apply

Apply a boundary condition to a node or face Group.

Clear

Remove a boundary condition from a node or face Group.

Fill Element Group



Material

Fill the selected element Group with a material.

Velocity

Assign a velocity to the selected element Group.

Initial Conditions

Assign initial conditions to the selected element Groups.

Transform group zoning

Translate

Translate (move) the current group.

Rotate

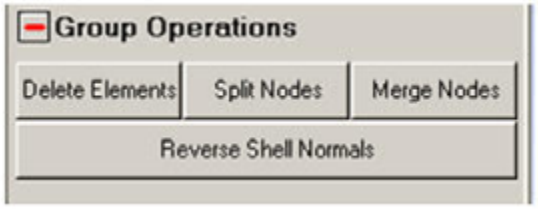
Rotate the current group.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Scale

Scale the current group.

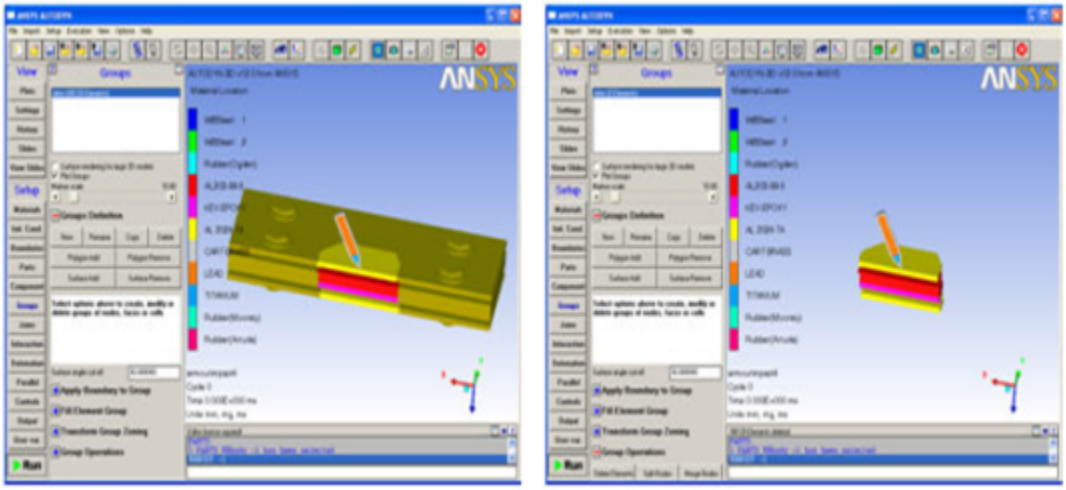
Group Operations



Delete Elements

Delete all elements within a selected Group. Select the Group containing the elements to delete and click **OK**. The elements are deleted from the model.

Figure 12.1: Example use of Delete Elements Group Operation



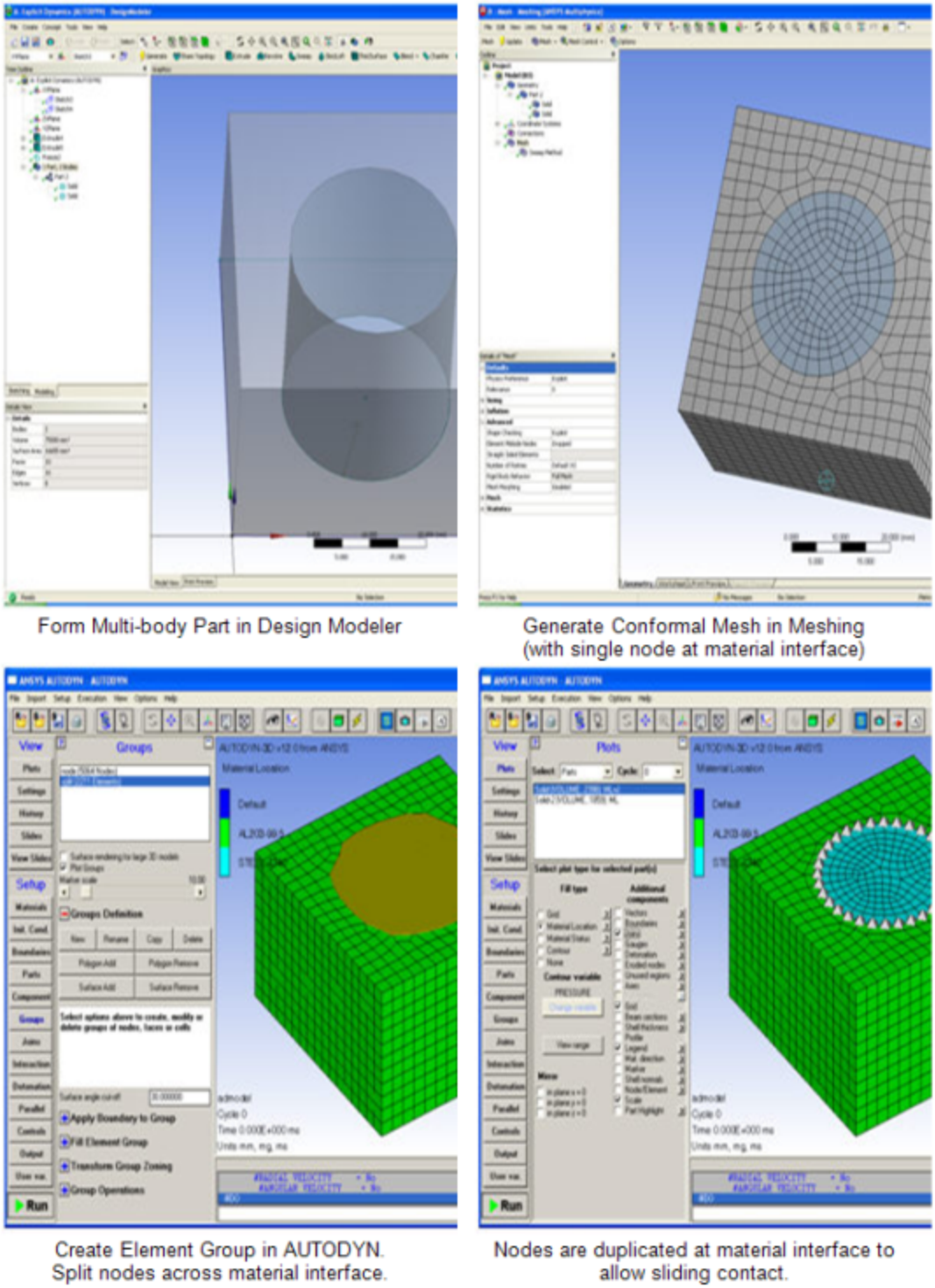
Split Nodes

Split nodes across material boundaries within a selected group. Autodyn will search through all elements in the selected group and identify nodes that are connected to elements (through the connectivity table) with different materials. Each node connected to more than one material will be replicated by the number of connected materials, while retaining the same physical location. The connectivity of each associated element will be updated.

The option is useful for generating a sliding interface between two materials and enforcing a conforming mesh (coincident nodes) between the material interface at the start of the simulation.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 12.2: Example use of Split nodes to create conformal sliding interface



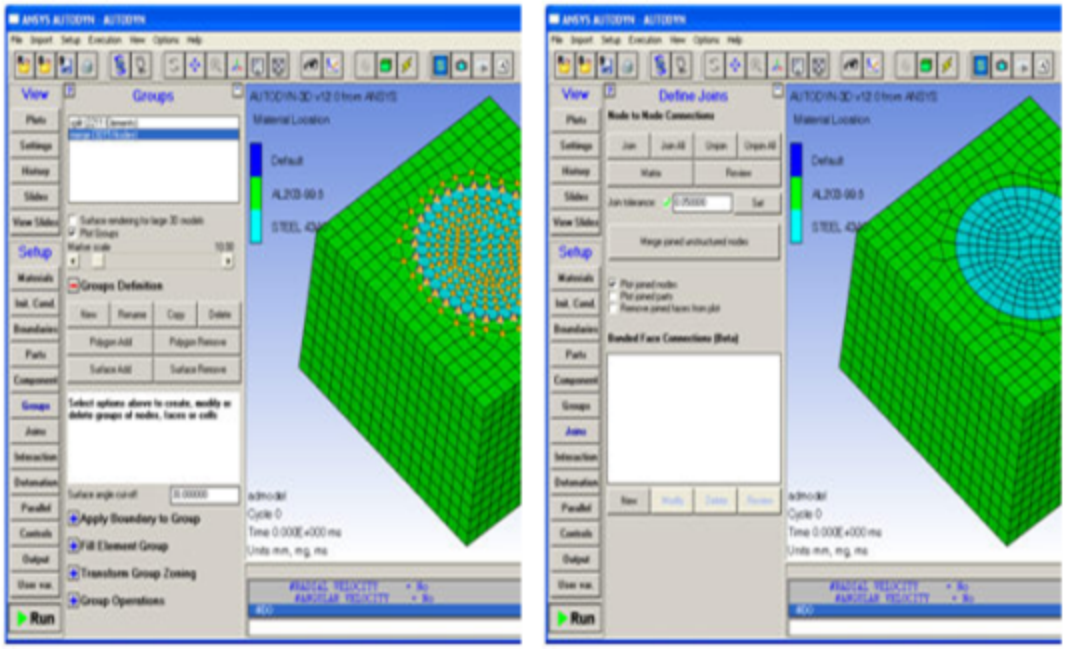
Merge Nodes

Merge coincident (joined) unstructured nodes into a single unstructured node. This option is similar to using the "Merged joined nodes" option on the Join panel; with this option only the nodes that are part of the node group will be merged.

This option will increase the robustness in many applications involving joins.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information
of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 12.3: Example use of Merge nodes to remove joins.



Create Node Group in AUTODYN.
Merge duplicate nodes.

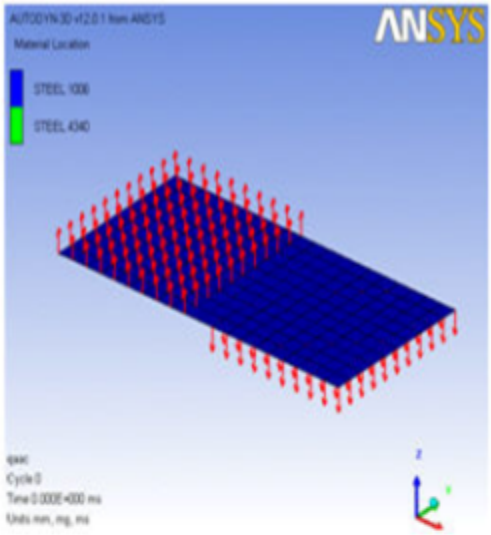
Duplicate nodes are merged into single nodes.

Reverse Shell Normals

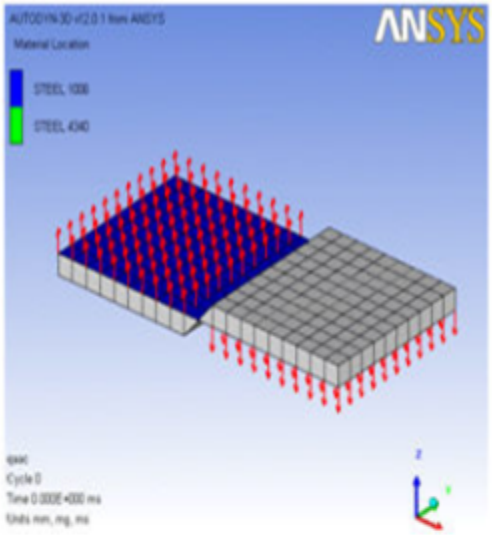
Reverse the shell element normal of a selected group of shell elements.

This option has to be used when not all element normals of a shell part point in the same direction and Euler-Lagrange coupling is used with thickened shells.

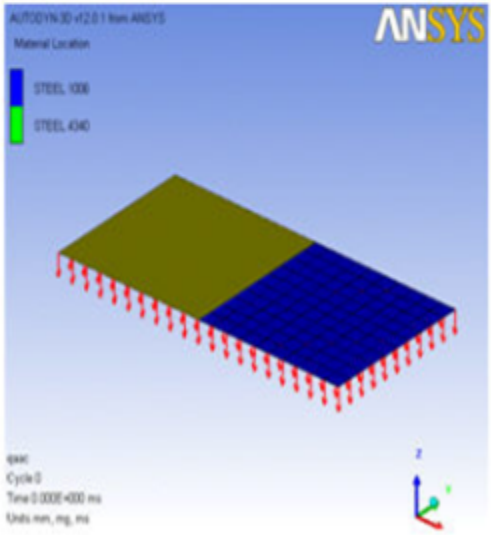
Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



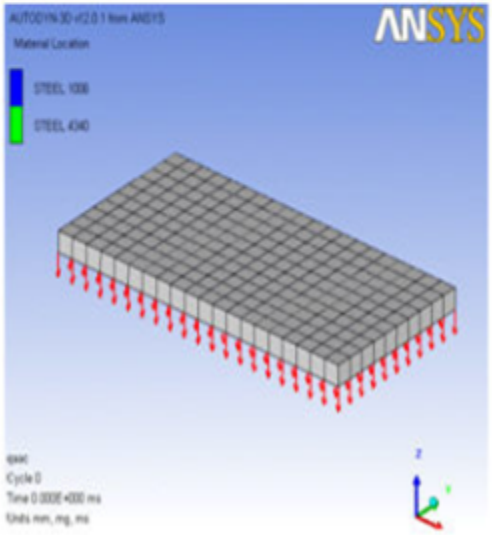
Single part with normals differently directed



Thickened shell provides incorrect coupling surface

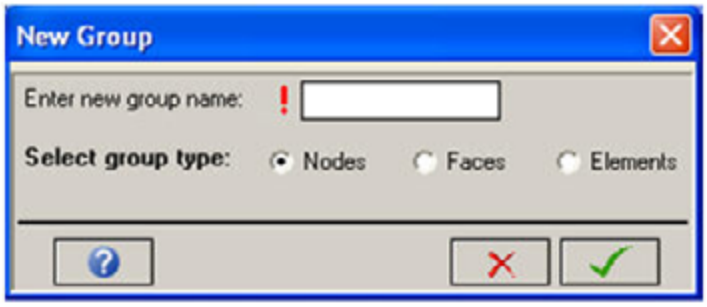


Create element group in AUTODYN Reverse normals



Thickened shell provides correct coupling surface

12.1. Groups - New



This window lets you create a Group.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Group Name

The Group name.

Group Type

Set either a node, face, or element Group type.

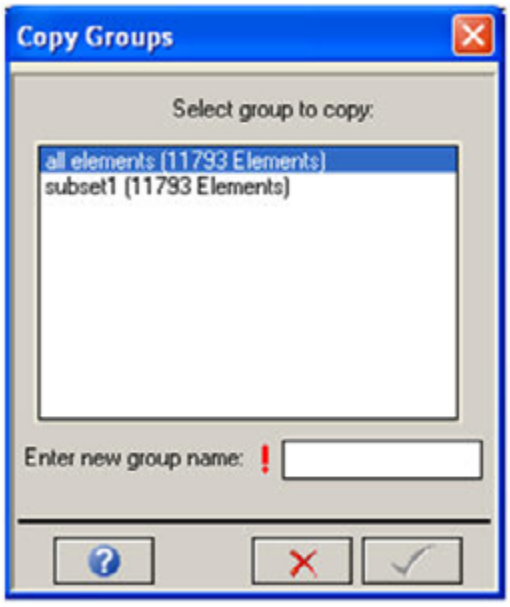
12.2. Groups - Rename

This window lets you rename an existing Group.

Group Name

The new Group name.

12.3. Groups - Copy



This window lets you create a copy of an existing Group.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

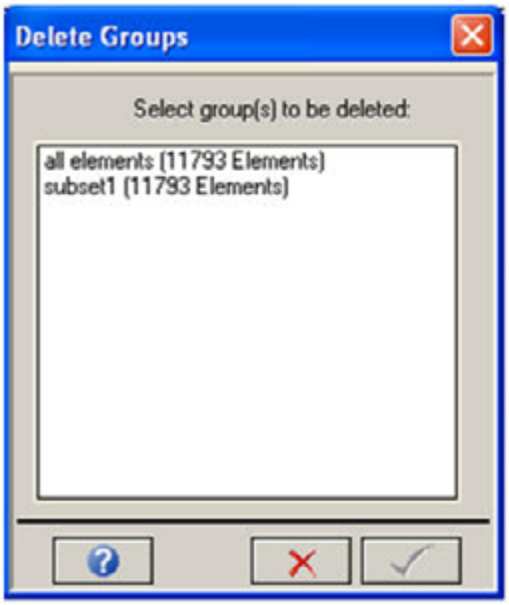
Group List

From the list, select the Group that you want to duplicate.

Group Name

Enter the new name for the copied Group.

12.4. Groups - Delete

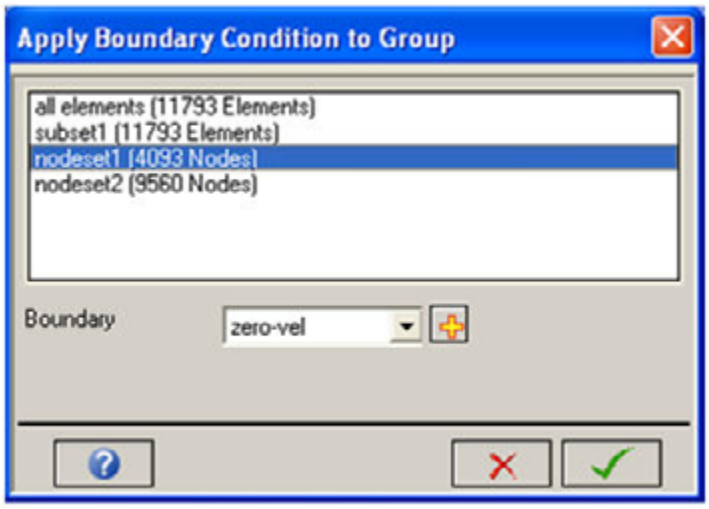


This window lets you delete one or more Groups.

Select the Groups you want to delete and click .

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

12.5. Groups - Apply



This window lets you apply a boundary condition to a Group. The type of Group and type of boundary condition must be consistent; that is, you can apply a nodal boundary condition only to a node Group, and a face boundary condition to a face Group.

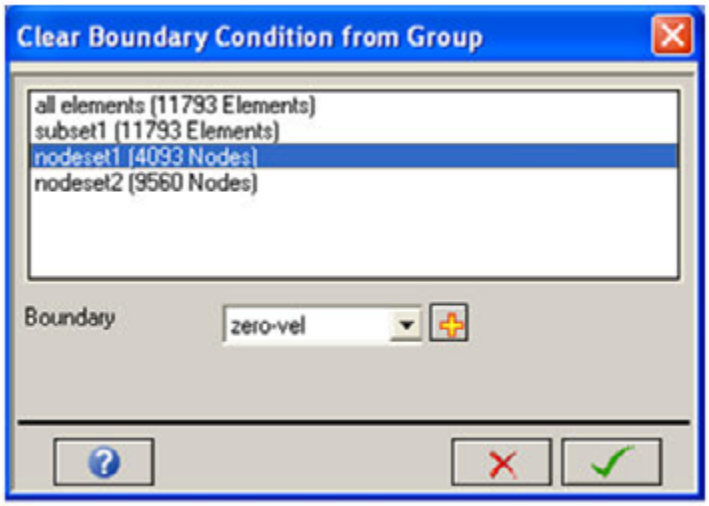
Group List

From the list, select the Group to which you want to apply the boundary condition.

Boundary

Select the boundary condition to be applied to the Group.

12.6. Groups - Clear



This window lets you clear a boundary condition to a Group. The type of Group and type of boundary condition must be consistent; that is, you can clear a nodal boundary condition only from a node Group, and a face boundary condition from a face Group.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

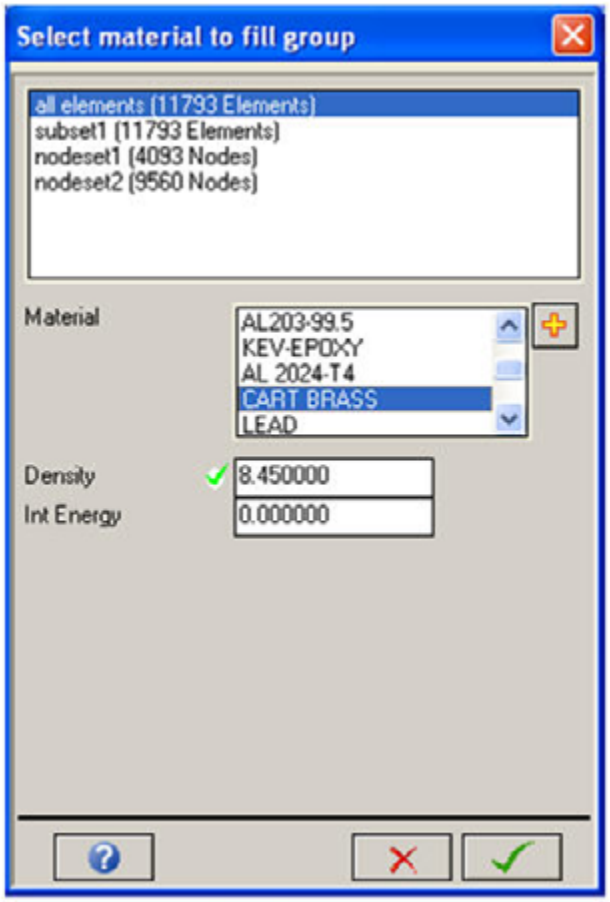
Group List

From the list, select the Group from which you want to clear from the boundary condition.

Boundary

Select the boundary condition to be cleared from the Group.

12.7. Groups - Material



This window lets you fill the selected element Group with a material.

Material

Select the material you want to use to fill your Group.

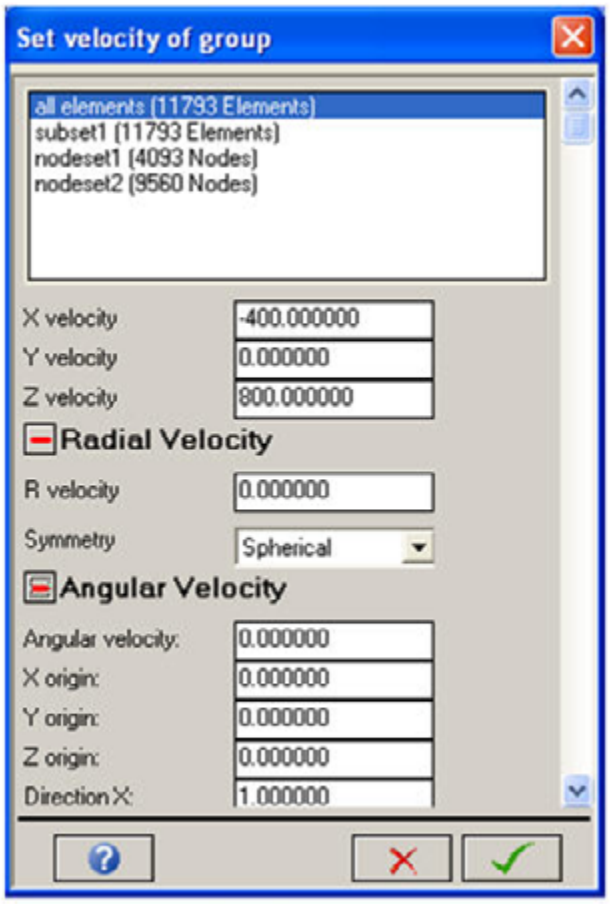
You can define Boundary Conditions by clicking the **Materials** button on the Navigation bar, or you can click next to the option in this window.

Material Parameters

Density and **Energy** values for your fill.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

12.8. Groups - Velocity

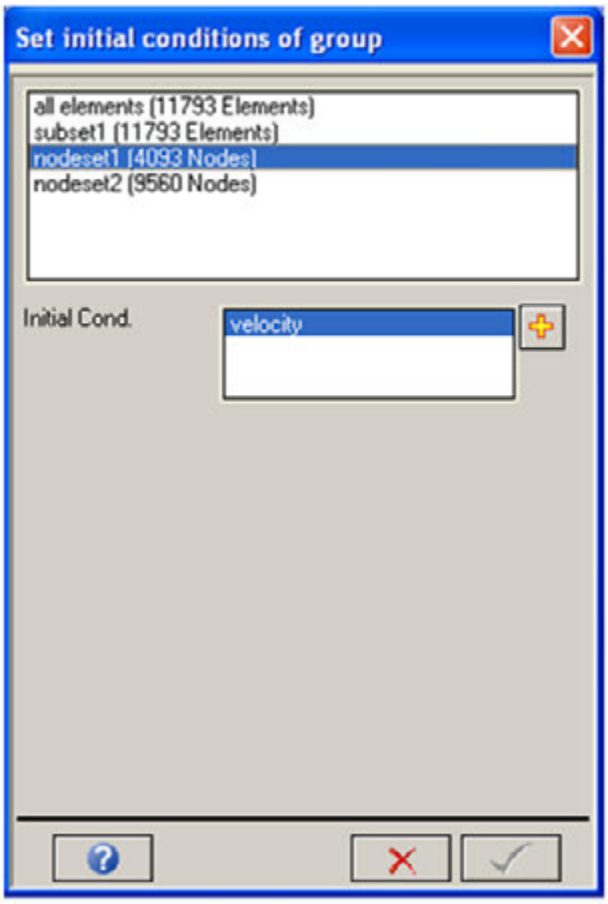


This window lets you assign a velocity to the selected element Group.

Enter the velocity components in the fields provided and click .

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

12.9. Groups - Initial Condition



This window lets you assign initial conditions from an Initial Condition Set to the selected element Group.

Fill with Initial Condition Set

Check this box to fill the Group using an Initial Condition Set.

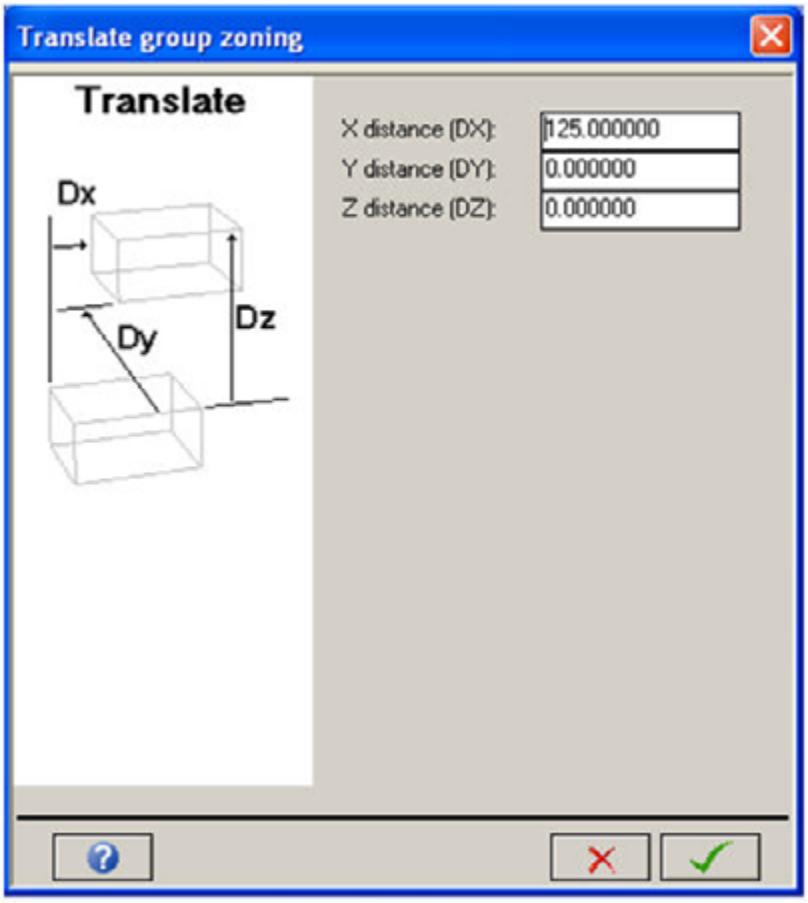
The advantage of filling using an Initial Condition Set is that if you later change one or more parameters in the set, you do not have to refill.

Use the pull-down menu to select the Initial Condition Set you want to use.

You can define new Initial Condition Sets by clicking the **Init.Cond.** button on the Navigation Bar or by clicking next to the option in this window.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

12.10. Groups - Translate



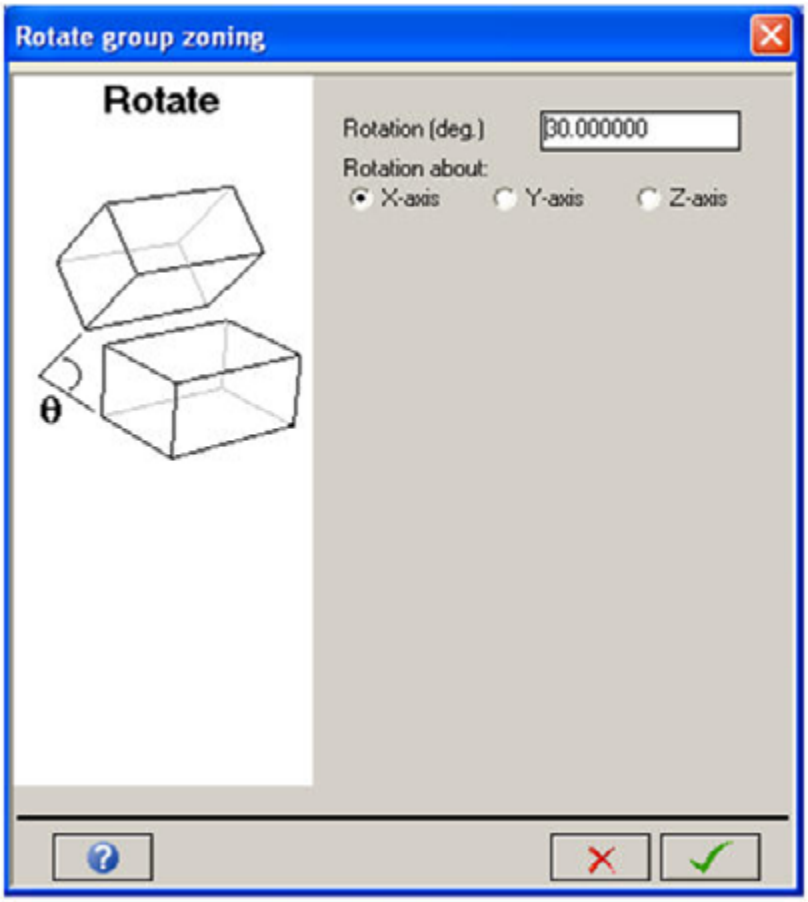
This window lets you translate all nodes associated with the current group in XYZ space.

X/Y/Z distance

The X, Y, and Z components of the translation.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

12.11. Groups - Rotate



This window lets you rotate all nodes associated with the current group about an axis in XYZ space.

Rotation

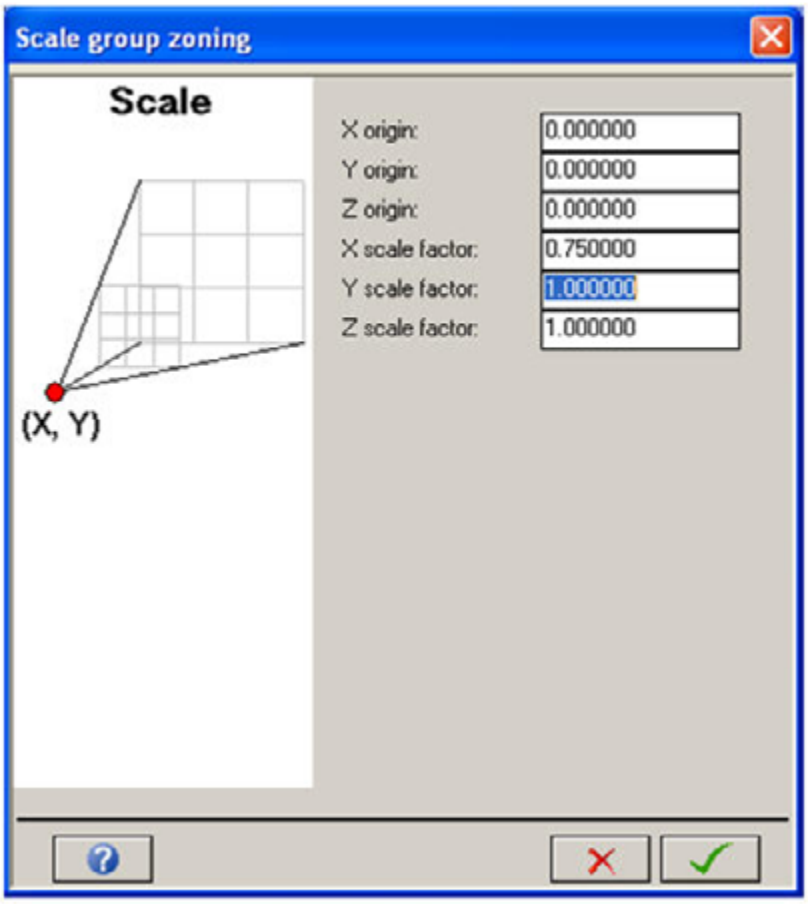
The rotation angle in degrees.

X/Y/Z-axis

The axis about which the nodes will be rotated.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

12.12. Groups - Scale



This window lets you scale all nodes associated with the current group relative to an origin in XYZ space.

Origin

The X, Y, and Z coordinates of the origin for the scaling.

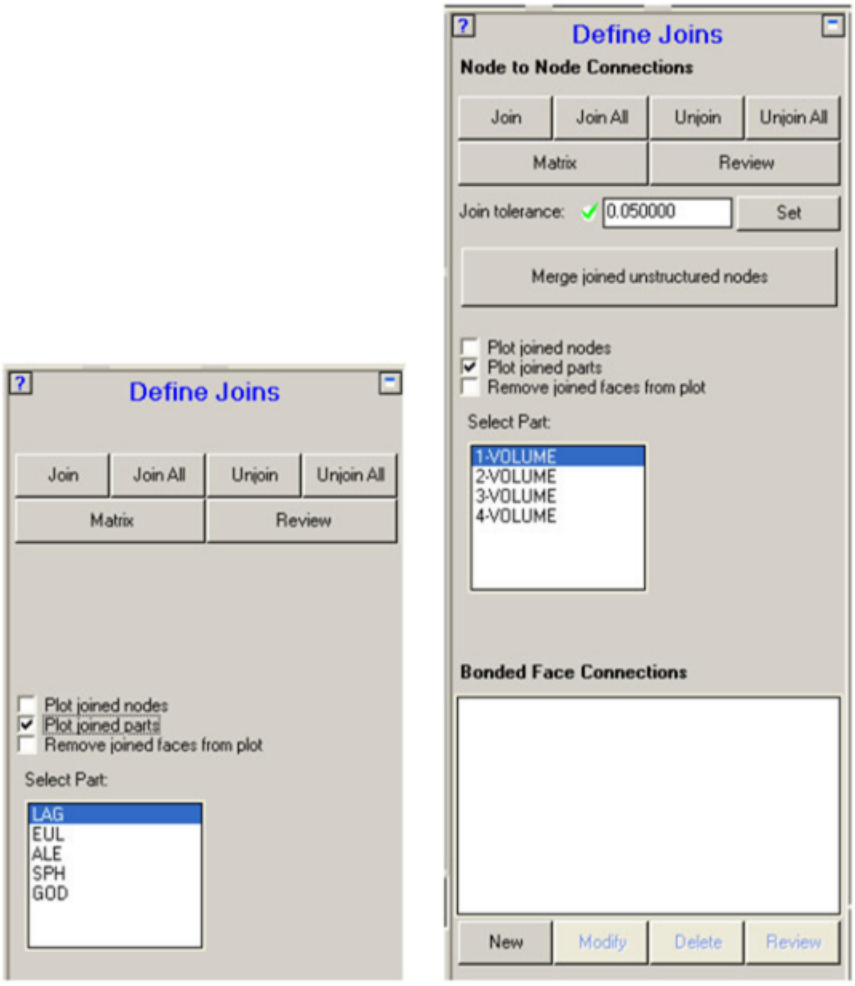
Scale Factors

The X, Y, and Z scale factors.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information
of ANSYS, Inc. and its subsidiaries and affiliates.

Chapter 13: Autodyn Joins



2D Window 3D Window

This panel lets you join Parts of your model.

If two Parts are joined, Autodyn automatically finds and joins all coincident nodes in the two parts.

Note:

Structured Lagrange elements joined to SPH nodes will not be eroded

---

When unstructured **Parts** are joined the coincident nodes can be merged into a single unstructured node. This option will increase the robustness in many applications involving joins.

Using the Bonded Face Connections option allows the nodes of an unstructured Part to be bonded to a face of another unstructured Part. Bonded contact alleviates the requirement that for joining purposes two nodes need to reside at the same physical location. The nodes that are to be bonded to a face may

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

have an offset with respect to the face and the offset is a user-defined maximum physical normal distance of a node from a face which can be bonded together.

Join

Joins Parts.

Join All

Join all Parts.

Unjoin

Unjoin Parts.

Unjoin All

Unjoin all Parts.

Matrix

Define join Parts using a matrix.

Review

Review which Parts are currently joined.

Join tolerance

Enter the tolerance you want Autodyn to use in determining which nodes should be joined. Click the **Set** button to set this tolerance.

The default value for the join tolerance will take care of small round-off differences in node location and will give good results. The tolerance should not artificially be increased to try to join nodes with a clear offset with respect to each other.

Merge joined nodes

Merges joined unstructured nodes into a single unstructured node.

Plot Joined Nodes

Check this box to plot a symbol at every joined node.

Plot Joined Parts

Check this box to show which Parts are joined to a selected Part (a box appears to let you select a Part).

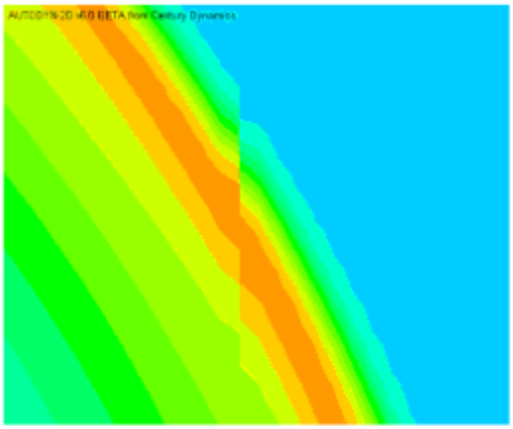
Remove joined faces from plot

Selecting this option results in continuous contours across joined faces and also removes joined faces from other types of plots.

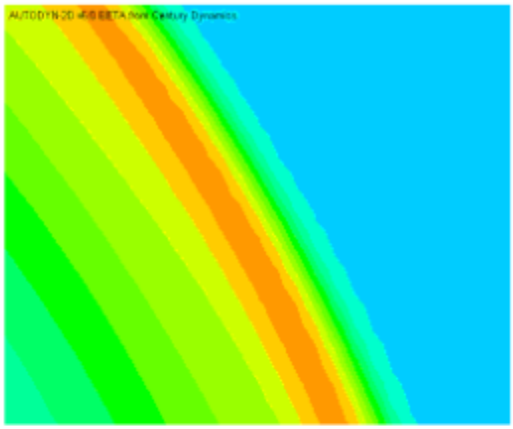
Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Improved Rendering Across Joined Nodes

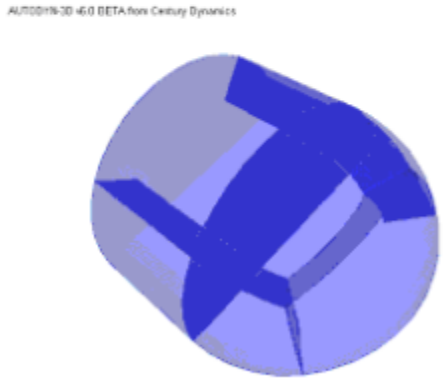
There is now an option to remove joined faces from rendered images under the Joins, Settings option. Selecting this option results in continuous contours across joined faces and also removed joined faces from other types of plots.



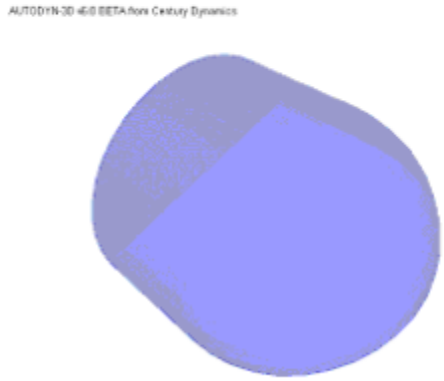
Contour plot including joined faces



Contour plot with "remove joined faces" set



Material plot using opacity



Material plot using opacity with "remove joined faces" set.

Bonded Face Connections

Bonded contact allows the nodes of an unstructured Part to be bonded to a face of another unstructured Part. Bonded contact alleviates the requirement that for joining purposes two nodes need to reside at the same physical location. The nodes that are to be bonded to a face may have an offset with respect to the face and the offset is a user-defined maximum physical normal distance of a node from a face which can be bonded together.

Bonded face connections can fail and break upon a user-defined stress criterion.

Spot Welds

Spot welds generated in an Ansys Explicit Dynamics systems are transferred to the Autodyn system as beam elements filled with a rigid material.

Spot welds utilize the multi-body rigid material option where each spot weld acts as an individual rigid boy.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Spot welds can be defined as breakable by force:

- Normal force limit: maximum normal force that can be sustained by the weld
- Shear force limit: maximum shear force that can be sustained by the weld.

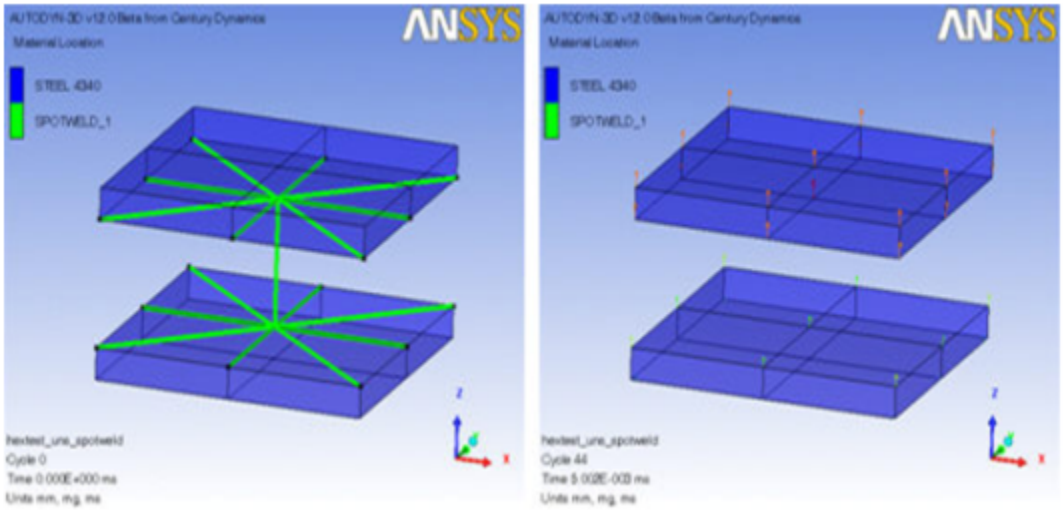
  Failure of the spot weld occurs when:

$$\left(\frac{|f_n|}{S_n}\right)^{\exp n} + \left(\frac{|f_s|}{S_s}\right)^{\exp s} \geq 1 \tag{13.1}$$

  where f and f are normal and shear interface forces, S and S are the maximum allowed normal and shear force limits and exp*n* and exp*s* are user-defined exponential coefficients. The normal interface force f is nonzero for tensile values only.

Note that spot welds cannot be generated directly inside an Autodyn system. Spot welds should be defined in an Ansys Explicit Dynamics system and automatically transferred to a linked Autodyn system.

For each spot weld connections a multiple beam rigid body is automatically generated for each spot weld in the Ansys Explicit Dynamics system in order to guarantee the transfer of bending over the spot weld.



After failure of the spot weld the spot weld rigid body will be removed from the analysis model as can be seen above.

Within an Ansys Explicit Dynamics system an effective area can be input for spot welds, thus the breakable criteria can be defined in terms of stress. This stress is converted to force when transferred into Autodyn.

Spot welds of zero length are permitted. However, if such spot welds are defined as breakable the above failure is modified since local normal and shear directions cannot be defined. A modified criterion using global forces is used instead:

$$\sqrt{\left(df_x^2 + df_y^2 + df_z^2\right) / \left(s_n^2 + s_s^2\right)} < 1.0 \tag{13.2}$$

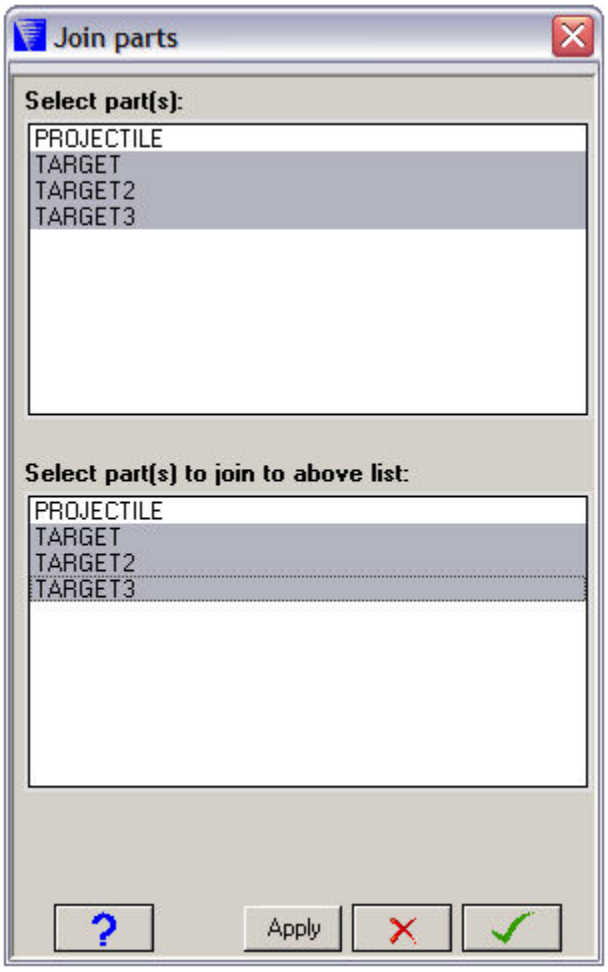
Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

where dfx, dfy and dfz are the global force differences across the spot weld.

Note:

- A spot weld is equivalent to a rigid body and as such multiple nodal boundary conditions cannot be applied to spot welds.
- Currently analysis models containing spot welds cannot be run in parallel.

13.1. Joins - Join



This window lets you join Parts.

This method of joining Parts works well no matter how many Parts you have in your model.

If you only have a few Parts, you may prefer to use a **Matrix** to choose the Parts you want to join.

Select Part(s)

Lists all the Parts in your model. Select one or more Parts in this list.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Select Part(s) to join to the above list

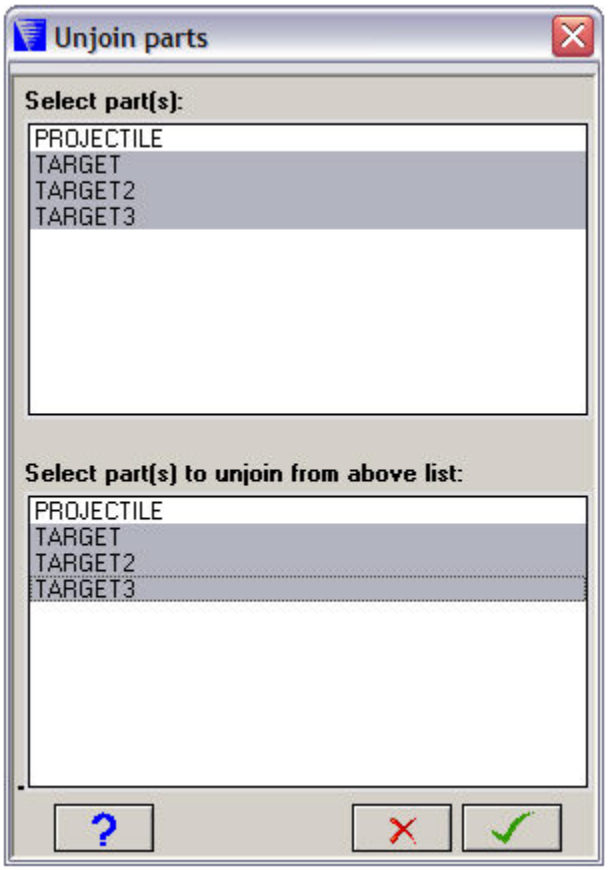
Also lists all the Parts in your model. Select one or more Parts in this list.

Apply

Click this button to join all Parts selected in the top window with all Parts selected in the lower window.

You can apply the above procedure as many times as you wish without exiting this window. Every time you click **Apply**, the specified Parts are joined.

13.2. Joins - Unjoin



This window lets you unjoin Parts.

This method of unjoining Parts works well no matter how many Parts you have in your model.

If you only have a few Parts, you may prefer to use a **Matrix** to choose the Parts you join (or unjoin)

Select Part(s)

This box lists all the Parts in your model. Select one or more Parts in this list.

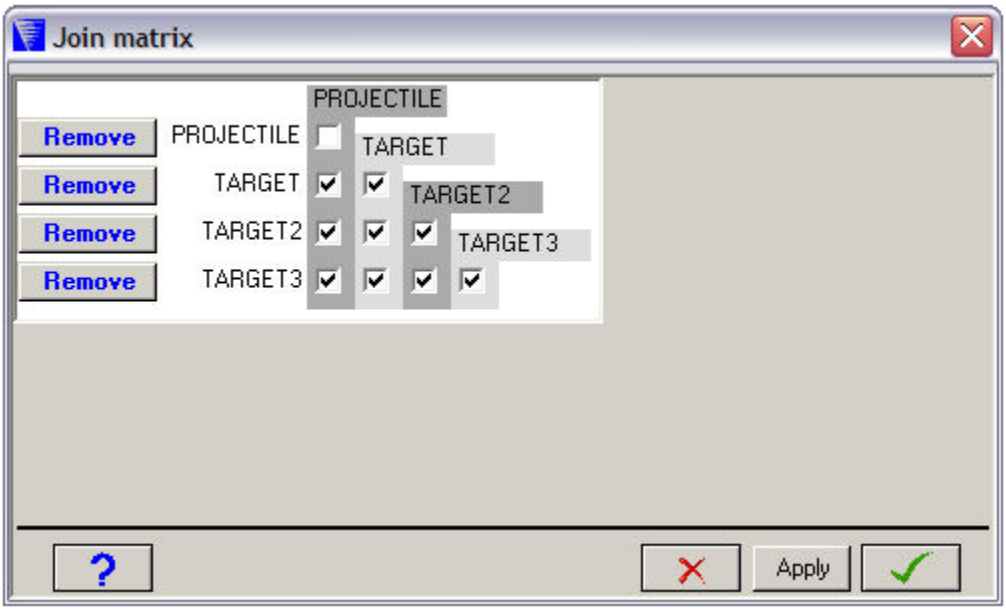
Select Part(s) to unjoin to the above list

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

This box also lists all the Parts in your model. Select one or more Parts in this list.

Press to unjoin all Parts selected in the top window with all Parts selected in the lower window.

13.3. Joins - Matrix



This window lets you choose the Parts you want to join by checking boxes in a matrix.

If you have a large number of Parts, you may find it easy to join / unjoin Parts using the **Join** / **Unjoin** options in the Joins dialog panel.

The matrix shown in this panel list the Parts of your model vertically and horizontally. This matrix is symmetrical, so only the diagonal and lower part of the matrix is shown.

Check a box to join the two associated Parts.

The Remove buttons let you quickly uncheck the boxes for each row of the matrix.

Press

to apply the changes you make to the matrix.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

13.4. Joins - Merge Joined Nodes

The option of merging joined unstructured nodes will merge the (multiple) joined nodes that reside at the same physical location in the model into one single unstructured node.

Since trajectory contact does not allow the use of joined unstructured nodes in the contact, the merge option can be used to remove joins from the model. Bonded connections are recommended over node to node joins for unstructured models.

13.5. Joins - Bonded Face Connections

Bonded contact allows the nodes of an unstructured Part to be bonded to a face of another unstructured Part. Bonded contact alleviates the requirement that for joining purposes two nodes need to reside at the same physical location. The nodes that are to be bonded to a face may have an offset with respect to the face and the offset is a user-defined maximum physical normal distance of a node from a face which can be bonded together.

Typically the regions to be bonded are selected by face groups or parts. This means that a bond can exist in a symmetrical fashion. The algorithm will try to find a face for all the nodes in the selected groups or parts.

It is also possible to select nodal groups. The nodes in these groups will also search for an appropriate face to be bonded to. Additionally the algorithm will deduce the faces that are attached to these nodes. This facilitates the bonding of nodal groups.

During the analysis the nodes are kept at the same relative position on the face to which they are bonded. This is done by means of penalty forces which are either dependent on the mass of the nodes/faces or the stiffness of the material. The stiffness is weighted based on materials on either side of the bond. In models with mass scaling the penalty method is chosen based on the mass scaling setting:

- Mass scaling off: Penalty method based on harmonic mass in the bonded pair.
- Mass scaling on: Penalty method based on harmonic stiffness in the bonded pair.

Origin of Model	Mass Scaling Off	Mass Scaling On
amodel_0.ad produced prior to R18.0	Harmonic Mass	Harmonic Mass

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

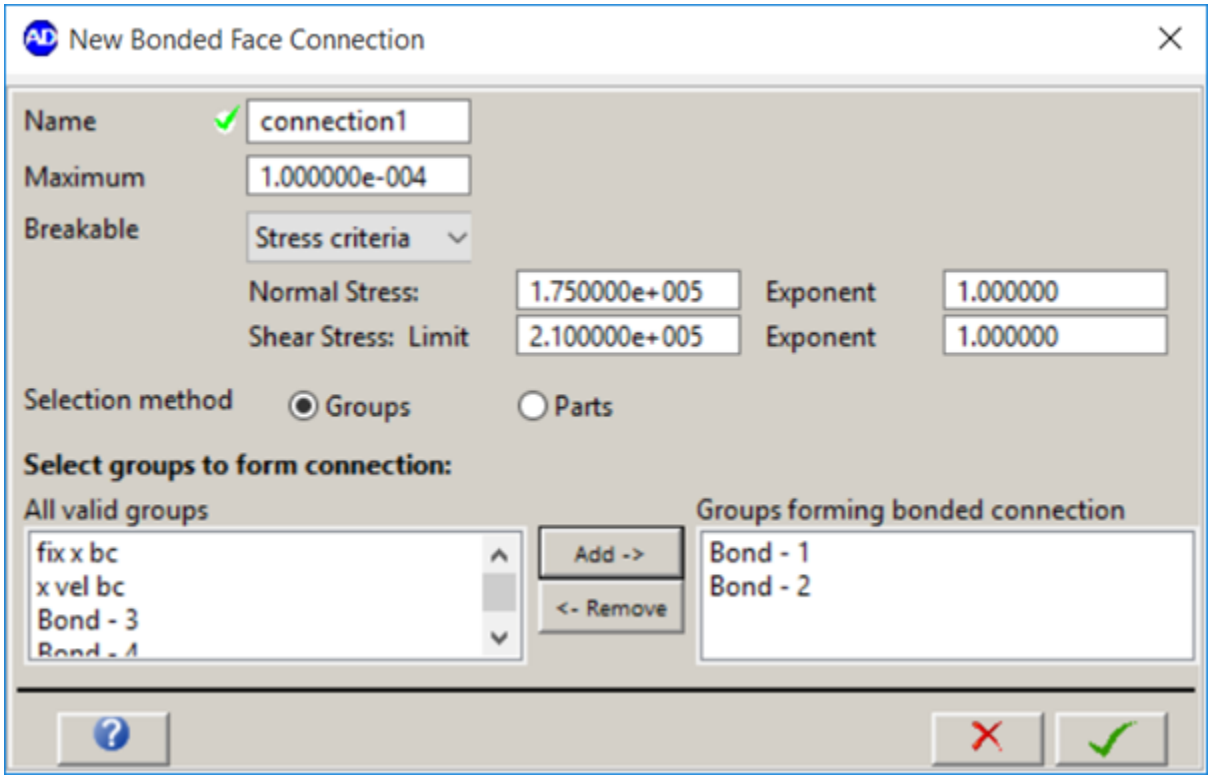
Origin of Model	Mass Scaling Off	Mass Scaling On
amodel_0.ad produced prior to R18.0 then saved in R18.0 or later	Harmonic Mass	Harmonic Stiffness
amodel_0.ad produced with current release		

Note:

The stiffness weighted penalty method is typically superior to a mass weighted penalty and increases the robustness of (offset) bonds. By switching on mass scaling and still using a small target timestep (eg 1e-20) no mass will be added, but the penalty method will be switched to harmonic stiffness.

When large material stiffness occurs between two materials that are bonded, it is recommended that you use an asymmetric definition where the contact scope (nodes to be bonded) refers to the soft material and the target scope (faces to bond to) refers to the stiffer material.

Bonds are allowed to fail and break upon a user-defined criterion.



Name

Enter a name for the bonded face connection set.

Maximum Offset

The automatic search algorithm for bonded regions will search for the minimum distance to any of the faces. If this minimum distance falls within the maximum offset value the bond pair will be es-

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

tablished. In order to compute the proper distance to a face the algorithm will determine if the perpendicular projection to the face falls within the face, if that is not the case the perpendicular projection to the face edges is considered, if that is not the case the distance to one of the face nodes is considered. This algorithm guarantees that a minimum distance is always found and can be properly compared against the user defined Maximum Offset.

Note:

In projects created before R18.0, models will still search for bonded pairs using a projection method. If these older models are saved they will start using the enhanced minimum distance search.

Breakable

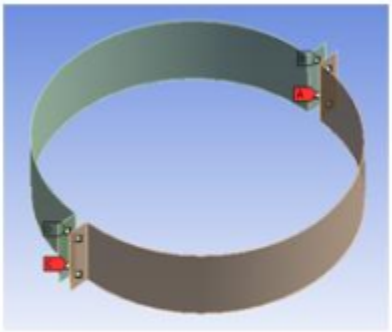
By default the bonded face connection will not be breakable. When the option Stress criteria is selected the bonded connection will fail and break upon a user-defined stress failure criterion:

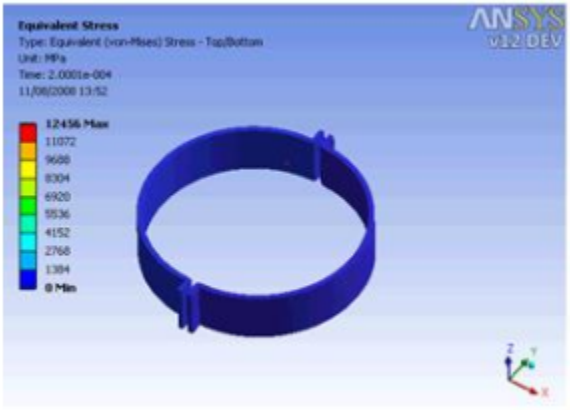
- Normal stress limit: maximum normal stress that can be sustained at the bond
- Shear stress limit: maximum shear stress that can be sustained at the bond.

Failure of the bond occurs when:

$$\left(\frac{|\sigma_n|}{S_n}\right)^{\exp n} + \left(\frac{|\sigma_s|}{S_s}\right)^{\exp s} \geq 1 \tag{13.3}$$

where σ_n and σ_s are normal and shear stress at the bond S_n and S_s are the maximum allowed normal and shear stress limits and exp_n and exp_s are user defined exponential coefficients. The normal stress at the bond σ_n is nonzero for tensile values only.





Selection method

Bonded joins can be generated in Autodyn on a part basis as well as on a (multiple) face and node group basis.

By default, the algorithm will attempt to bond all the nodes and faces in the definition in a symmetric fashion. This behavior can be changed if you define the bonds by groups. In order to facilitate anti-symmetric bonding behavior in the Autodyn system, the following naming convention can be used:

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Group Name starts with: BONDED_CONTACT-

This name indicates a contact side; all nodes in this group will be part of the node list to be bonded. None of the associated faces in this group will be a possible target for bonding.

Group Name starts with: BONDED_TARGET-

This name indicates a target side; all the associated faces in this group will be a possible target for bonding. None of the nodes in this group will be bonded.

Group Name starts with: BONDED_SYM-

Note that the only way to reset the behavior of a group to symmetric once the Contact and Target side names are used, is to use BONDED_SYM-.

Note:

- Bonded contact is available for unstructured volume, shell, and beam elements.

13.5.1. Verification of Bond Status

You can verify the status of the bonds in your analysis in several different ways, outlined in the following sections.

13.5.1.1. Using the Prt File Output

13.5.1.2. Using Preprocessing Output

13.5.1.3. Using Post Processing Output

13.5.1.4. Bonded Contact Warning Messages

13.5.1.1. Using the `Prt` File Output

Verification of the initialized bonds can be done by inspection of the `prt` file. A summary is given which lists the number of candidate nodes for bonding and the actual number of nodes that were bonded. If the percentage of nodes to be bonded is 0% it means none of the nodes are actually bonded. Please consider increasing the Maximum Offset in this case.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

```
BONDED CONTACT SUMMARY

  Bonded Contact algorithm: Harmonic Stiffness

                                                                  Number of Bonded Nodes
  Bond Name                 Max Offset    Breakable    Behaviour   Candidates       Actual    Bonded  Internal Bond Name
  Bonded - PRT0003_adh     1.00000E-01          No    Symmetric         3823         2913    76.20%  BONDED -    1
  Bonded - MOTUSA_ITO1     8.10000E-01          No    Symmetric         5335         2380    44.61%  BONDED -    2
  Bonded - ITO To MOTU     7.00000E-01          No    Symmetric         7601         7289    95.90%  BONDED -    3
  Bonded - ITO To SCRE     1.00000E+00          No    Symmetric         7608         7296    95.90%  BONDED -    4
  Bonded - Multiple To     5.00000E-01          No    Symmetric          279          167    59.86%  BONDED -    5


  Scope Definition:
  Bonded - PRT0003_adh
    Internal name: BONDED -    1
    Group(s):
                          BONDED- 2                          1205 Faces
                          BONDED- 1                          3533 Faces
  Bonded - MOTUSA_ITO1
    Internal name: BONDED -    2
    Group(s):
                          BONDED- 4                          3616 Faces
                          BONDED- 3                          1205 Faces
  Bonded - ITO To MOTU
    Internal name: BONDED -    3
    Group(s):
                          BONDED- 6                          3677 Faces
                          BONDED- 5                          3616 Faces
  Bonded - ITO To SCRE
    Internal name: BONDED -    4
    Group(s):
                          BONDED- 8                          3677 Faces
                          BONDED- 7                          3630 Faces
  Bonded - Multiple To
    Internal name: BONDED -    5
    Group(s):
                          BONDED-10                           316 Faces
                          BONDED- 9                            96 Faces
```

13.5.1.2. Using Preprocessing Output

In order to visualize the bonds it is possible to use the following advanced procedure. Please note that this is only available in the Autodyn component system and that a backup of your model file at cycle zero needs to be made because beam elements are added to the model to visualize the bonded connections.

1. Define and set the environment variable:

   ATD_GROUPPLOT_BONDED_CONTACT = ON

2. Start Autodyn and open the model (make sure you do this after defining the environment variable).

3. Save your model as, for example, <admodel_db_0.ad>

4. Run the model for 1 cycle.

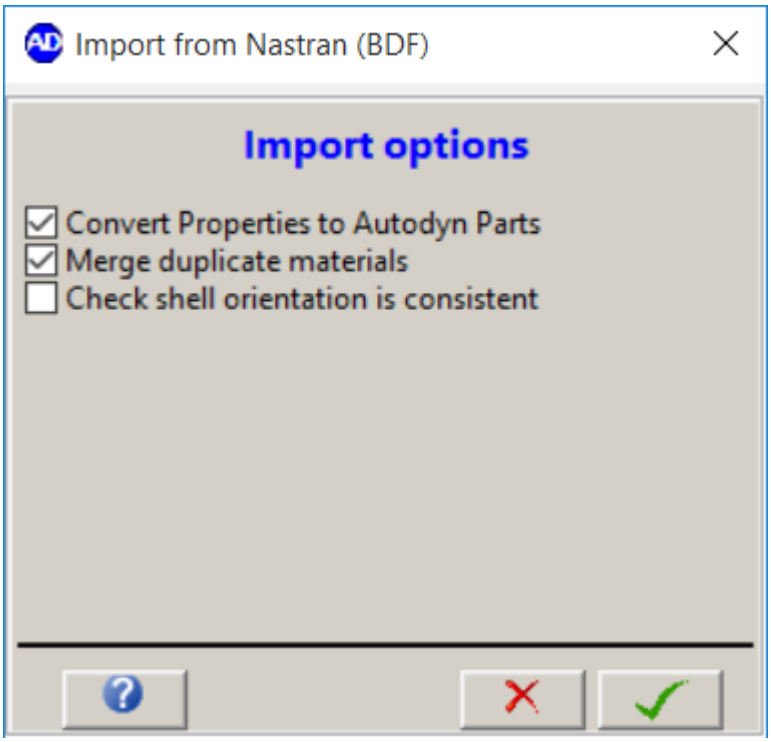
   This will generate output like:

   admodel-db_BONDED-1_NCYC-0_NCONNECT-1.dat
   admodel-db_BONDED-2_NCYC-0_NCONNECT-2.dat

5. Go back to cycle 0 and import the files via menu: Import -> Import from Nastran (BDF).

6. During import, make sure you select the checkbox **Convert Properties to Autodyn Parts**.

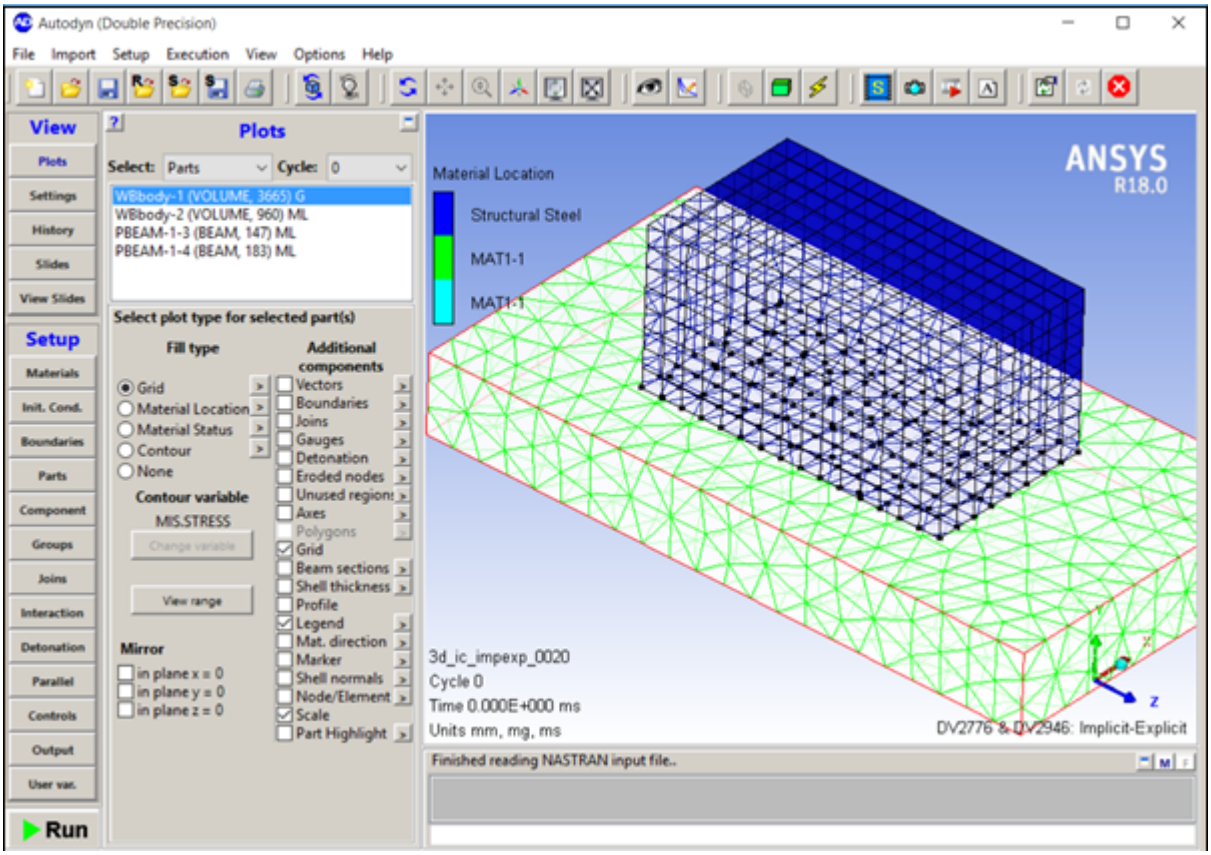
Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



7. You will now have a new part containing the definition of the bond used in the simulation. If the bonds are not offset it is best to visualize them by activating the Grid tick box under Additional Components and by reducing Opacity of the corresponding parts.

8. In the example pictured here, the black dots visualize the bonds which are the end nodes of tiny beams in this case. Note the two new parts that were created upon import of the `dat` files.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



13.5.1.3. Using Post Processing Output

Additional verification can be done via the element variable BOND_STATUS. It is used to record how many nodes are actively bonded to the faces of an element. This variable can be contoured to verify that bonds are correctly defined. If a bond fails during the simulation the bond status will show a reduction in the number of active bonds. When an element has no remaining bonds the value of BOND-STATUS is set to -1.0.

13.5.1.4. Bonded Contact Warning Messages

During initialization of the bonded connections the solver will do some definition checks because a *candidate* node can only be bonded to one target face. The solver does allow a face to be the target for multiple nodes, which makes it possible to always resolve the warnings. The warnings may not always be problematic, but it is wise to inspect the model setup to make sure the definition was done properly. Two checks are in place:

- A candidate node can exist in multiple bonded connection definitions. Ultimately the solver will only pick the closest face to the node and disregard all the other faces. If a node is a candidate in multiple definitions the user will be warned in the log-file with the message: "There are *n* bonded connection(s) that contain the same node(s) in the scoped Contact geometry."

  An example of this is a plate that is bonded asymmetrically between two solid geometries. The nodes of the plate will only bond to one side. In this case it is wise to also (or only) bond the nodes of the solid to the plate. Bonding the nodes of the solid geometries only asymmetrically to the plate will most likely yield the most robust results and will resolve the warning message.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- By default, the maximum offset is defined as 1e-4 mm. This allows bonding when geometry faces are coinciding. In cases where (small) offsets are existing between the faces, this maximum offset will be too small to generate any bonds. If this situation occurs, you will be warned in the log file with a message: "There are *n* bonded connection(s) where none of the Candidate nodes found a target face."

  Typically this situation can be resolved by increasing the maximum offset. Also, it is wise to double check the scope of both the Contact and Target definition. You may have selected the same face or edge in both the contact and target scope, which results in no bonded contact pairs. A third item to check is the overall definition of all bonded connections. If candidate nodes in the contact scope list are also used in other bonded connections they may take precedence over the definition at hand. In the latter case the message will always co-exist with the message in the first bullet. Defining the bonded connections asymmetrically gives you full control over the bonding definition and will resolve the warning messages in the latter case.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information
of ANSYS, Inc. and its subsidiaries and affiliates.

Chapter 14: Autodyn Interactions

Introduction

The Interactions dialog panel lets you define interactions between the various Parts of your model.

Buttons at the top of this panel let you choose which type of interaction you want to set up:

Lagrange/Lagrange

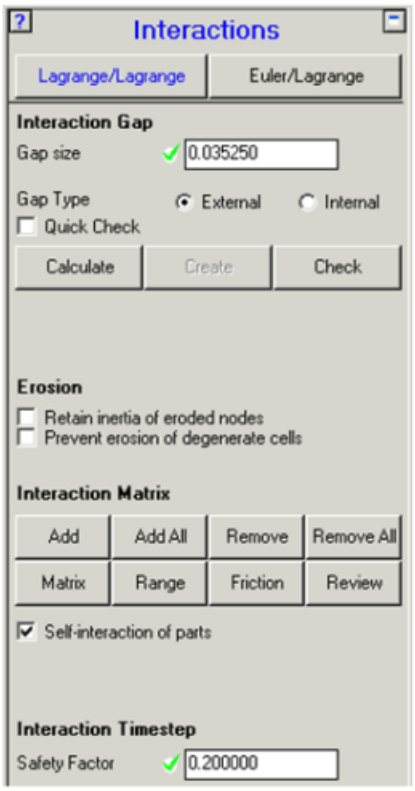
Click this button to set up impact / slide interactions between Lagrangian Parts (Parts using a Lagrange, Shell, Beam or SPH solver).

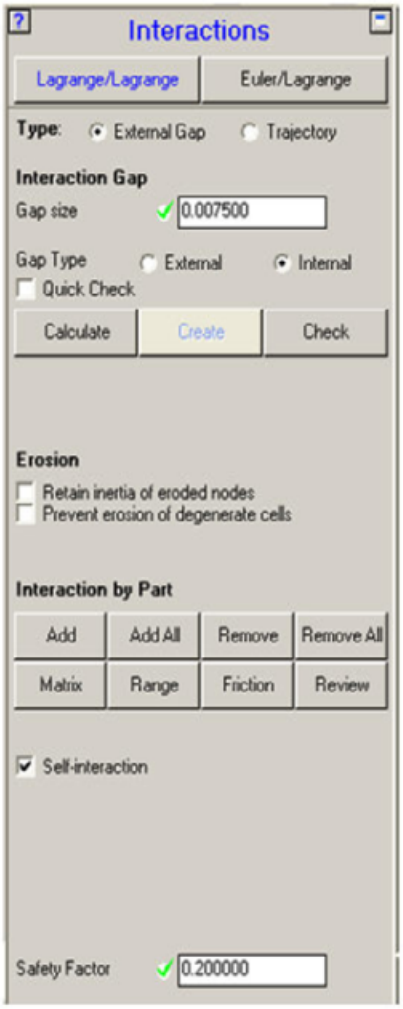
Euler/Lagrange

Click this button to set up coupling between Lagrangian Parts and Eulerian Parts.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

14.1. Interactions - Lagrange/Lagrange





This panel lets you set up interactions between Lagrangian Parts (Parts using a Lagrange, Shell, Beam or SPH solver).

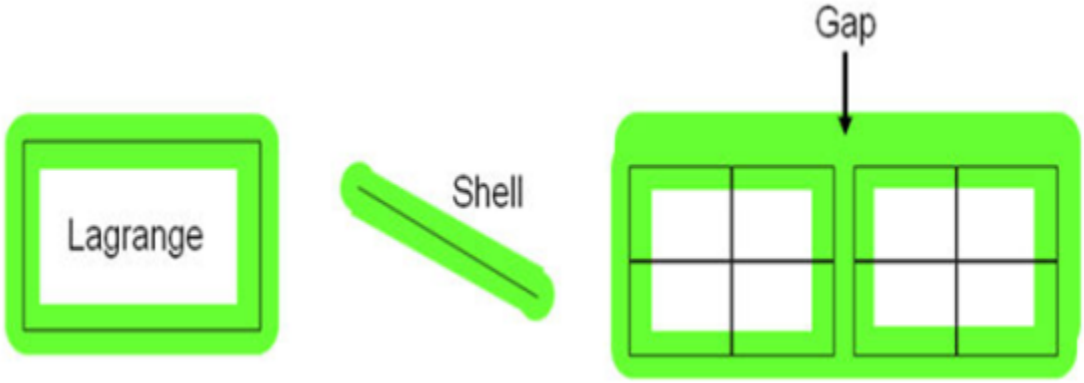
14.1.1. Interactions - Lagrange/Lagrange - Introduction

Autodyn offers 2 types of Lagrange/Lagrange interaction (contact) methods.

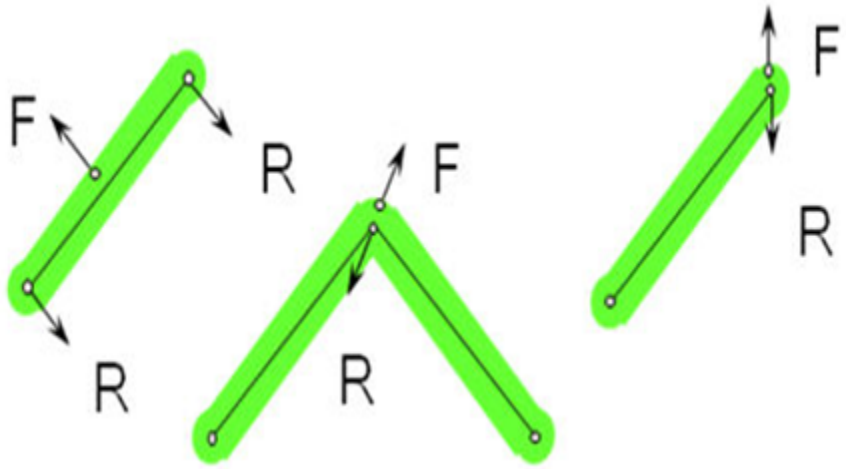
- **Gap contact**

  In this contact method each surface segment is surrounded by a contact detection zone. The dimensions of this contact zone is called Gap size. This means that initially the outside surfaces of the parts in the contact should be separated by the gap distance.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



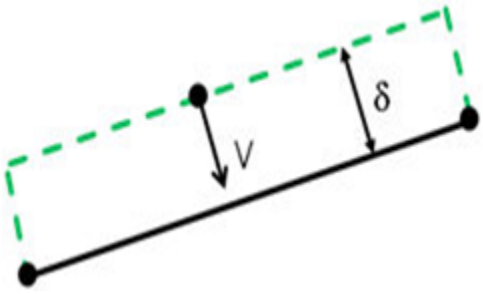
Any nodes entering the contact detection zone of a surface segment are repelled by a force normal to the segment surface and proportional to the depth of penetration of the node into the contact detection zone.



The interaction conserves linear and angular momentum.

To assure a stable interaction process the gap contact algorithm uses a timestep restriction such that during 1 computational time step a surface node cannot travel more than 20% into the contact detection zone.

$$\Delta t = 0.2\frac{\delta}{V}$$



where δ is the gap size and V the velocity of the penetrating node.

The gap contact is available for all structured as well as unstructured Lagrange Parts, with exception for 2D unstructured Parts.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- **Trajectory contact**

  The Trajectory contact algorithm is available for all 3D Unstructured solvers, and has three main potential benefits over the Gap based contact algorithm.

  – There is no requirement to specify a contact detection zone or leave a physical gap between Parts at the start of a simulation. This makes model generation for complex 3D geometries significantly easier.

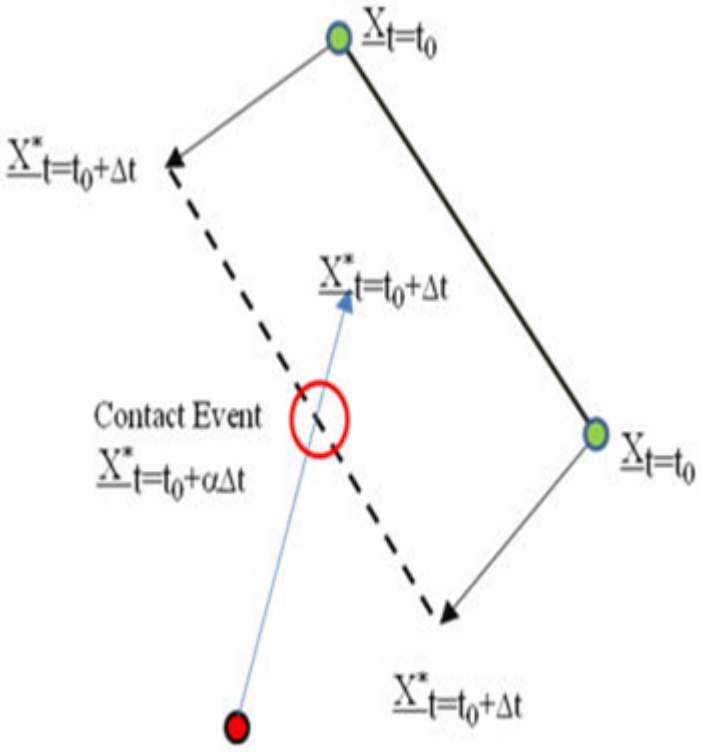
  – There is no constraint on the timestep due to contact. The algorithm detects node to face contact by tracking the trajectory of the nodes and faces over time. The removal of the timestep constraint on contact can give very significant performance improvements.

  – The algorithm is energy conserving, in addition to momentum conserving.

  The trajectory of nodes and faces included in frictional or frictionless contact are tracked during the computational cycle. If the trajectory of a node and a face intersects during the cycle a contact even is detected.

  Note:

  The use of SPH parts with Trajectory Contact is not supported. To use SPH parts with Trajectory Contact (in a serial solution only), please contact Ansys Technical Support.



If a contact event is detected the node involved in the contact will be pushed back towards the true contact position during the computational cycle.

There are two methods available:

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

1. **Penalty**

   If a contact event is detected, a local penalty force is calculated to push the node back to the face. Equal and opposite forces are calculated on the nodes of the face in order to conserve linear and angular momentum. The applied penalty force will push the node back towards the true contact position during the cycle. However, it will take several cycles to satisfy the contact condition. Kinetic energy is not necessarily conserved. The conservation of energy can be tracked using the energy time history.

2. **Decomposition Response (DCR)**

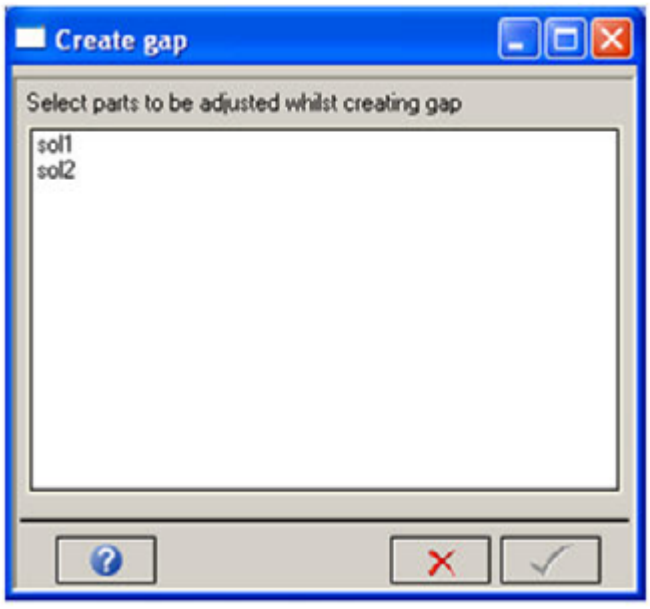
   All contacts that take place at the same point in time are first detected. The response of the system to these contact events is then calculated to conserve momentum and energy. During this process, forces are calculated to ensure that the resulting position of the nodes and faces does not result in further penetration at that point in time. The decomposition response algorithm is more impulsive (in a given computational cycle) than the penalty method. This can give rise to large hourglass energies and energy errors.

The trajectory contact can handle multiple contact of a node with different faces of a contact surface during a computational timestep.

Current Limitations

- The trajectory contact option cannot be used in combination with joined unstructured parts. To circumvent this limitation the option to merge all joined unstructured nodes can be used.
- Interaction of SPH nodes, accounting for their full natural thickness/size is currently not available.
- The decomposition response algorithm cannot currently be used in combination with bonded contact regions. The formulation will automatically switch to the Penalty method if bonded regions are present in the model.

14.1.2. Interactions - Lagrange/Lagrange - Create



Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

This window lets you create a gap between structured Parts automatically. This option does not work for unstructured parts and therefore unstructured Parts will not appear in the window.

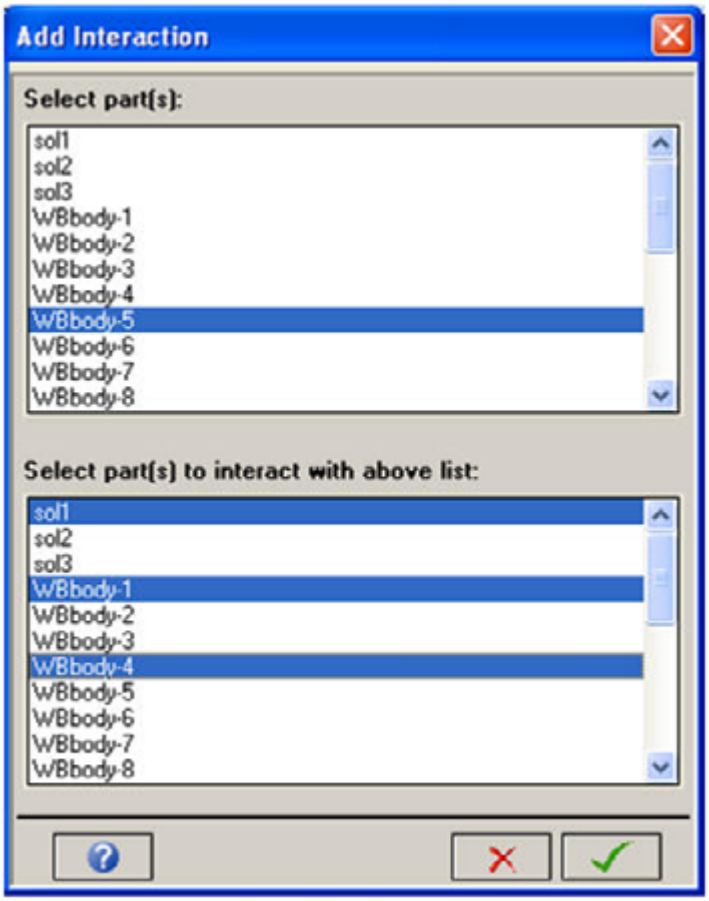
Gaps are created by reducing the size of surface elements of the selected Parts, so there will be small changes in the mass of the Parts that are modified.

You may prefer to manually inset gaps using the **Translate** and / or **Scale** options in the **Zoning** panel (no mass will be lost).

The box in this window lists all Parts involved in interactions.

Select the Part(s) you want to adjust while creating the gaps and click for Autodyn to automatically create the gaps.

14.1.3. Interactions - Lagrange/Lagrange - Add



This window lets you add interactions between one or more Parts.

This method of adding interactions works well no matter how many Parts you have in your model.

If you only have a few Parts, you may prefer to use a **Matrix** to choose the interactions.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Select Part(s)

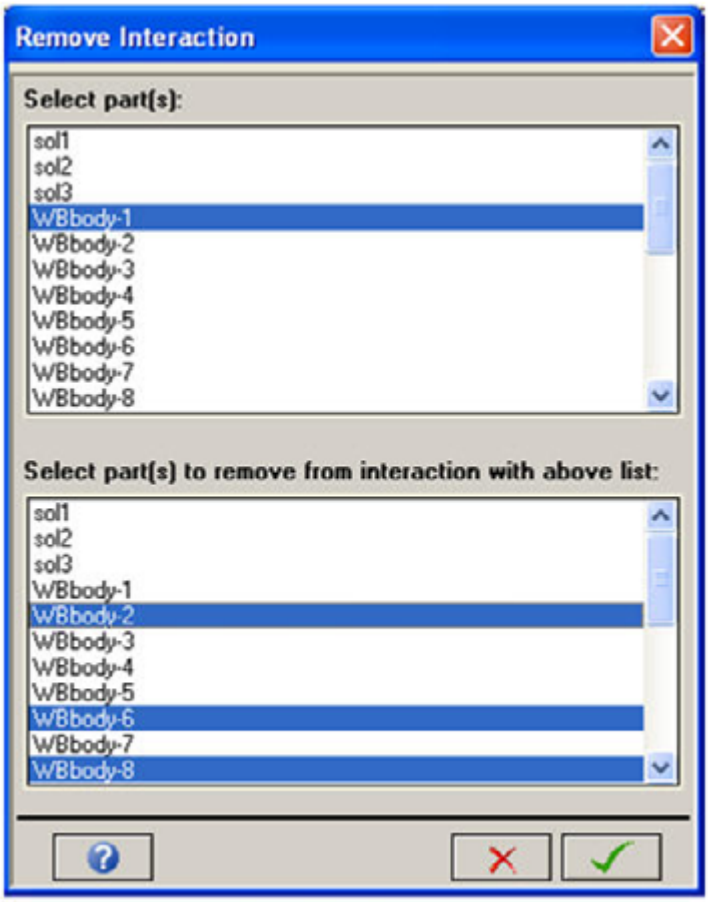
This lists all Parts involved in interactions. Select one or more Parts in this list.

Select Part(s) to interact with the above list

This also lists all Parts involved in interactions. Select one or more Parts in this list.

Click to set all Parts selected in the top window to interact with all Parts selected in the lower window.

14.1.4. Interactions - Lagrange/Lagrange - Remove



This window lets you remove interactions between one or more Parts.

This method of removing interactions works well no matter how many Parts you have in your model.

If you only have a few Parts, you may prefer to use a **Matrix** to add or remove interactions.

Select Part(s)

This lists all Parts involved in interactions. Select one or more Parts in this list.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

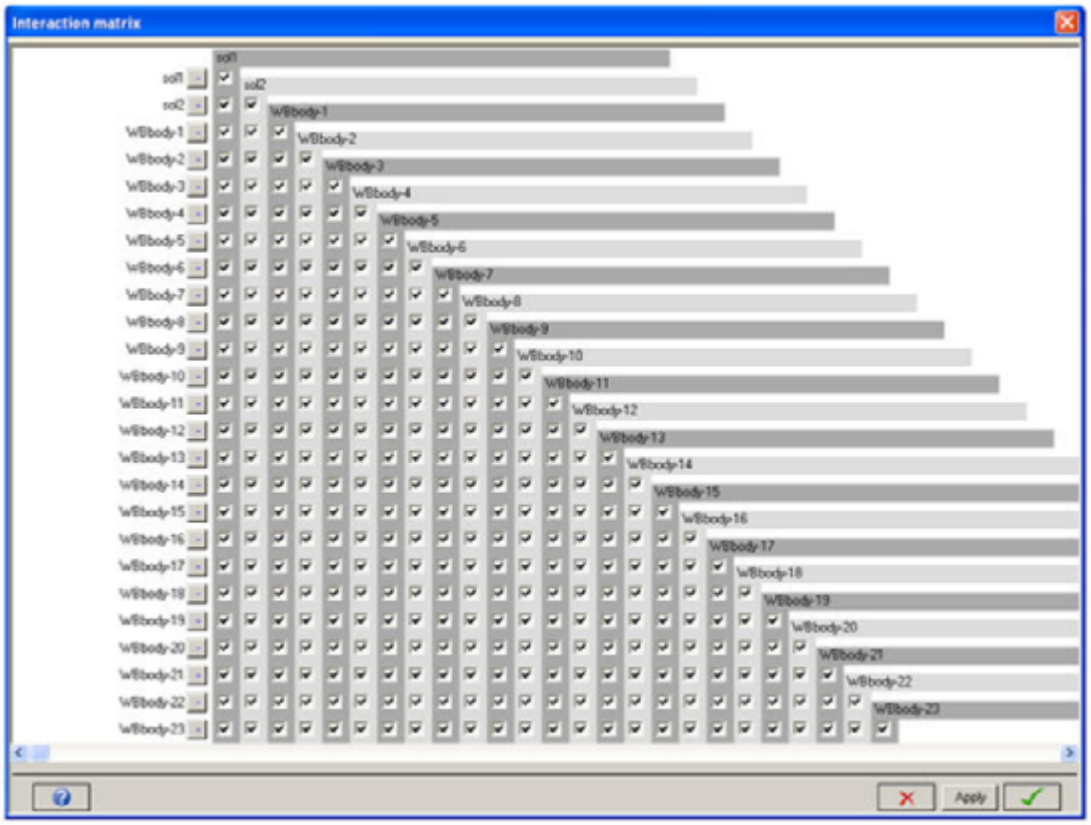
Select Part(s) to interact with the above list

This also lists all Parts involved in interactions. Select one or more Parts in this list.

Select Part(s)

Click to remove interaction between all Parts selected in the top window and all Parts selected in the lower window.

14.1.5. Interactions - Lagrange/Lagrange - Matrix



When the model does not contain more than 25 parts, this window lets you choose which Parts you want to interact by checking boxes in a matrix.

When a model contains more than 25 parts, the matrix option is not available and will be greyed-out.

If you have a large number of Parts, you may find it easy to set interactions using the **Add** / **Remove** options in the **Interactions** dialog panel.

The matrix shown in this panel lists the Parts of your model vertically and horizontally. This matrix is symmetrical, so only the diagonal and lower part of the matrix is shown.

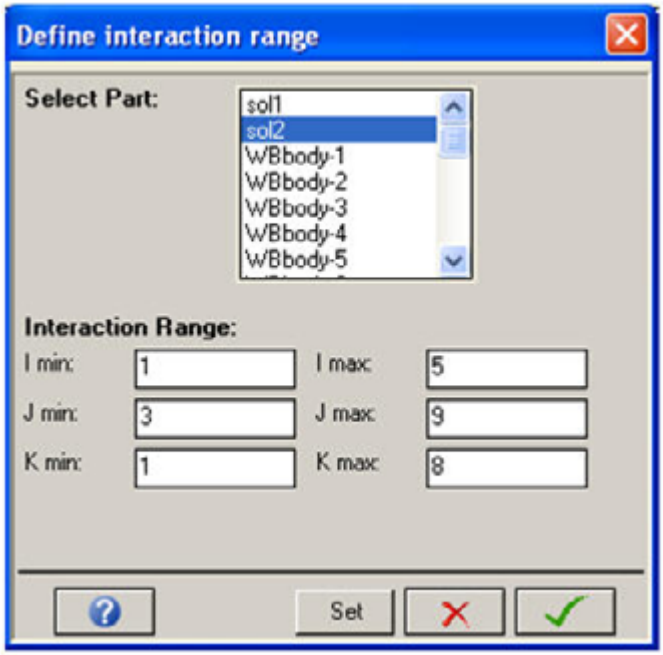
Check a box if you want the two associated Parts to interact.

The Remove buttons let you quickly uncheck the boxes for each row of the matrix.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Click to apply the changes you make to the matrix.

14.1.6. Interactions - Lagrange/Lagrange - Range



This window, available only for Gap contact, lets you set the index range of a structured Part that will be checked for interactions. This option is not available for unstructured parts.

For Trajectory contact and for unstructured parts the Interaction by Group option can be used instead to select only a region of a Part for interaction.

Select Part

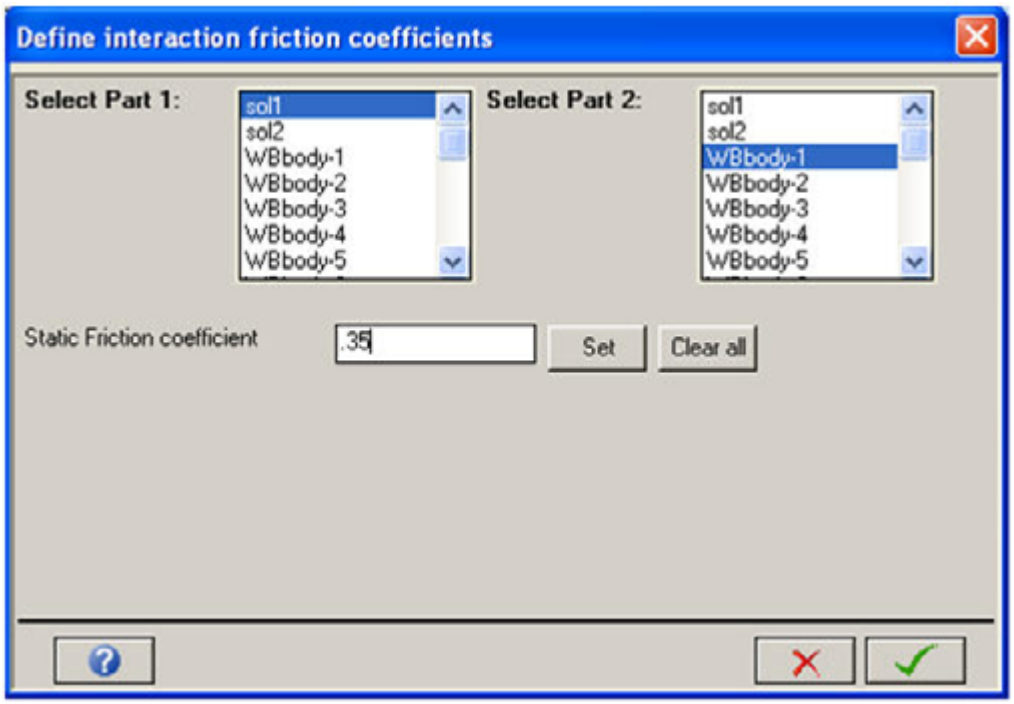
Select the Part for which you want to set the index range for interactions in this box.

Interaction Range

Enter the index range that you want to be checked for interaction.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

14.1.7. Interactions - Lagrange/Lagrange - Friction



This window is available for Gap contact and lets you set static friction coefficients between two Parts.

Select Part 1

Select the first of the two Parts you want to set the friction coefficient between in this box.

Select Part 2

Select the second of the two Parts you want to set the friction coefficient between in this box.

Friction Coefficient

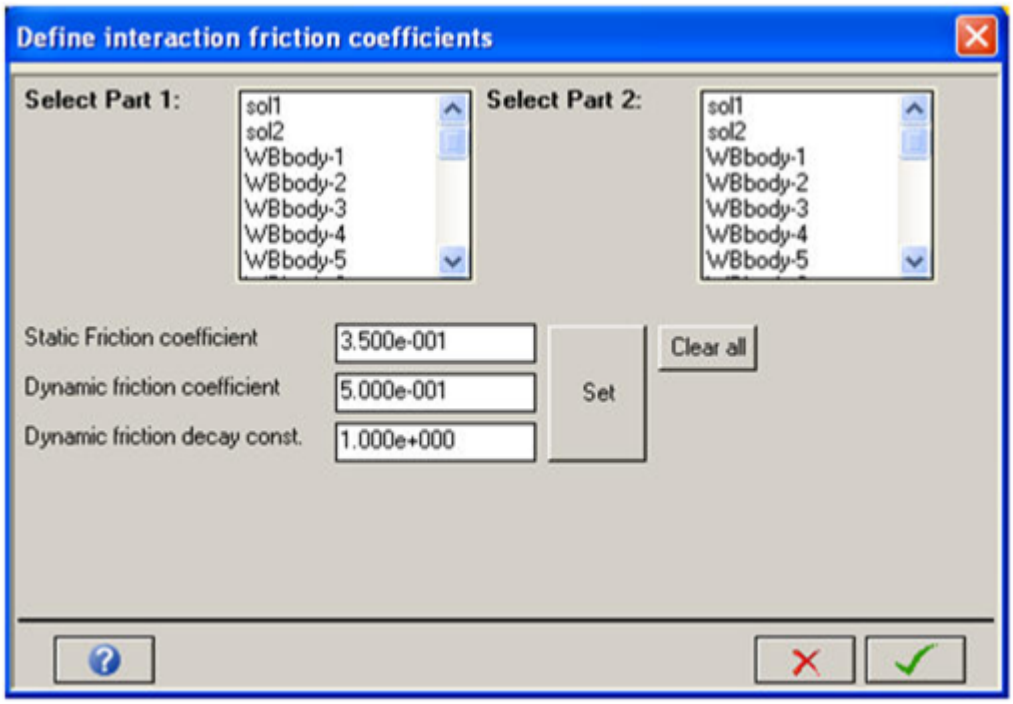
Enter the friction coefficient you want to use.

Click

to set the friction coefficient between the two selected Parts.

You can apply this procedure as many times as you like to set coefficients between other Parts without exiting the window.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



This window is available for Trajectory contact and lets you define static and dynamic friction between two Parts.

Select Part 1

Select the first of the two Parts you want to set the friction coefficient between in this box.

Select Part 2

Select the second of the two Parts you want to set the friction coefficient between in this box.

Friction Coefficient

- A non-zero value for the Static Friction coefficient will activate Coulomb type friction between bodies (F=μR).
- A non-zero value for the Dynamic Friction coefficient and the Dynamic friction decay constant will activate the use of dynamic friction where an effective friction coefficient, μ, is used which is given by:

$$\mu=\mu_d+\left(\mu_s-\mu_d\right)e^{-\beta v} \tag{14.1}$$

where:

μ_s = Static

μ_d = Dynamic coefficient of friction

β = Exponential decay coefficient

v = Relative sliding velocity at point of contact

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Click to set the friction coefficient between the two selected Parts.

You can apply this procedure as many times as you like to set coefficients between other Parts without exiting the window.

Note:

When using gap contact, friction is not available for Interaction defined by Group. Friction is available when using group contact with trajectory contact. See Interactions - Lagrange/Lagrange - 3D Trajectory Contact Options (p. 423) for information on how to define friction coefficients when using group contact with trajectory contact.

14.1.8. Interactions - 2D Gap Contact Options

Type - External Gap

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

When a 2D model contains structured elements only, the 2D gap contact must be used to define Lagrange/Lagrange interactions.

An analysis model that contains a mix of structured (IJK) Lagrangian Parts and unstructured Lagrangian Parts can currently not be run with Gap contact and also not with Trajectory contact.

To be able to use Trajectory contact the analysis model must contain unstructured Lagrangian volume Parts only. Similar, to be able to use Gap contact the analysis model must contain structured Lagrangian Parts only.

Interaction Gap

A gap is required for the interaction logic. The size of this gap must be in the range 1/10 to 1/2 the dimension of the smallest element face of Parts involved in interactions.

For most models you should set the gap size close to the smallest allowed value (Parts using a Lagrange or Shell Solver).

- **Gap size**

  Enter a value for the Gap size in this field or click **Calculate** to have Autodyn set it for you.

- **Gap Type**

  Select whether you want to use an **internal** or **external** gap.

  The external gap is *strongly* recommended because calculations using this option are more robust and more efficient. However, this option does require that interacting Parts are separated by the gap size at the start of your calculation.

- **Quick check**

  Check this box to use a fast calculation for the interaction time-step using only the maximum velocity in your problem. You should only check this box if you know that your maximum velocities will always be in impact regions.

- **Calculate**

  Click this button to have Autodyn automatically calculate the Gap size for you. The Gap size will be set to 1/10 the dimension of the smallest element face of Parts involved in interactions.

- **Check**

  Click this button to check that the Gap size is valid and that all Parts involved in interactions are separated by the Gap size.

Erosion

Retain inertia of eroded nodes

Check this box if you want to retain the inertial of eroded nodes (otherwise eroded nodes are removed from the model).

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Prevent erosion of degenerate elements

Check this box if you want the interaction calculation to be used to stop elements from becoming degenerate. This option only works for Lagrange volume elements.

> **Warning:**
>
> Activating this option can lead to unphysical results, particularly if a large Gap size is used.

Interaction by Part

Add

Sets interactions between Parts.

Add All

Sets all Parts to interact with each other.

Remove

Removes interactions between Parts.

Remove All

Removes all interactions between Parts.

Matrix

Sets interactions using a Part matrix.

Range

Sets the index range of each Part that will be checked for interactions.

Friction

Sets static friction coefficients between different Parts.

Review

Enables a review of all of your interaction settings.

Self Interaction

If the self-interaction option is switched on the contact detection algorithm will also check for external nodes of a part contacting with faces of the same part in addition to other parts. This is the most robust contact setting since all possible external contacts should be detected.

Interaction Timestep

Autodyn automatically calculates an Interaction Timestep based on the Gap size and the maximum velocity of the Parts involved in interactions.

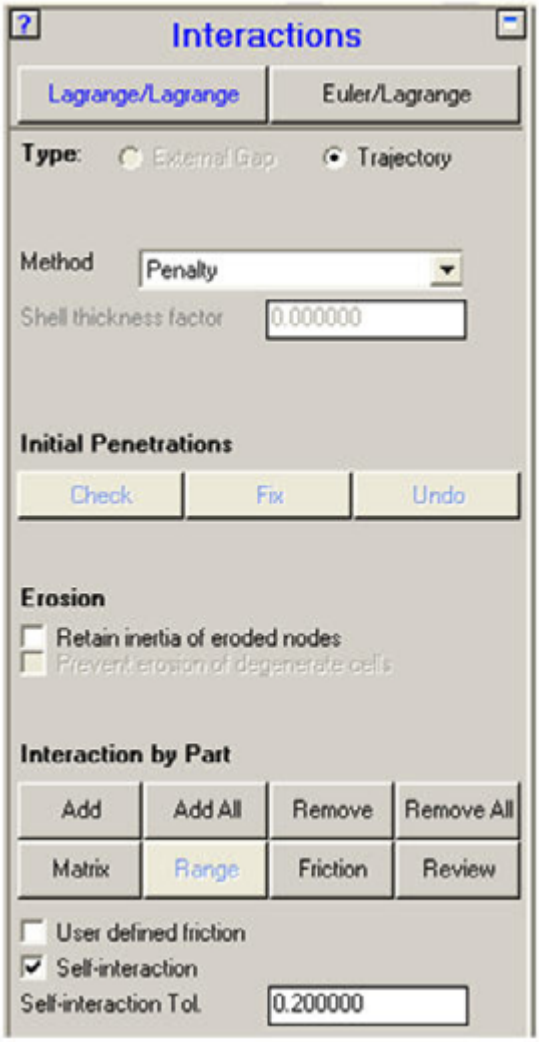
Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Safety Factor

The safety factor you want to use to calculate the Interaction Timestep.

You cannot enter a value greater than the default value (0.2).

14.1.9. Interactions - Lagrange/Lagrange - 2D Trajectory Contact Options



Type Trajectory

When a 2D model contains unstructured Lagrangian volume elements the 2D trajectory must be used to define Lagrange/Lagrange Interactions.

An analysis model that contains a mix of structured (IJK) Lagrangian Parts and unstructured Lagrangian Parts can currently not be run with Trajectory contact and also not with Gap contact.

To be able to use Trajectory contact the analysis model must contain unstructured Lagrangian volume Parts only. Similar, to be able to use Gap contact the analysis model must contain structured Lagrangian Parts only.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Method

Method in which a contacting node is pushed back to the true contact position during the computational cycle.

There are three methods available:

1. **None**

   No contact defined

2. **Penalty**

   If a contact event is detected, a local penalty force is calculated to push the node back to the face. Equal and opposite forces are calculated on the nodes of the face in order to conserve linear and angular momentum. The applied penalty force will push the node back towards the true contact position during the cycle. However, it will take several cycles to satisfy the contact condition. Kinetic energy is not necessarily conserved. The conservation of energy can be tracked using the energy time history.

3. **Decomposition Response (DCR)**

   All contacts that take place at the same point in time are first detected. The response of the system to these contact events is then calculated to conserve momentum and energy. During this process forces are calculated to ensure that the resulting position of the nodes and faces does not result in further penetration at that point in time. The decomposition response algorithm is more impulsive (in a given computational cycle) than the penalty method. This can give rise to large hourglass energies and energy errors.

Retain inertia of eroded nodes

Check this box if you want to retain the inertial of eroded nodes (otherwise eroded nodes are removed from the model).

Interaction by Part

Add

Sets interactions between Parts.

Add All

Sets all Parts to interact with each other.

Remove

Removes interactions between Parts.

Remove All

Removes all interactions between Parts.

Matrix

Sets interactions using a Part matrix.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Friction

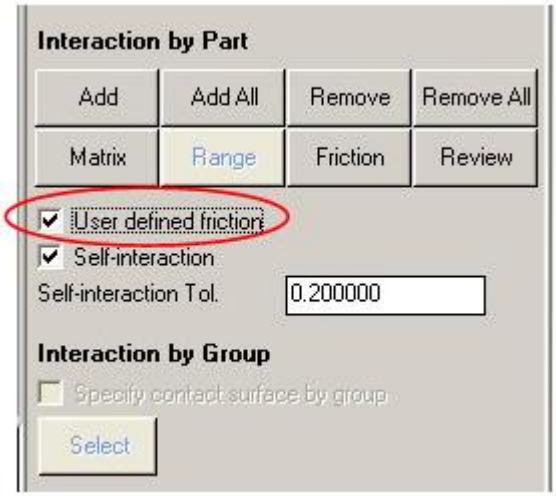
Sets static friction coefficients between different Parts.

Review

Enables a review of all of your interaction settings.

User defined friction

Aside from the friction definition through the Interaction by Part method described above there is also a user defined friction option available through the use of a user-subroutine called EXFRICTION2D.



Self Interaction

If the self-interaction option is switched on the contact detection algorithm will also check for external nodes of a part contacting with faces of the same part in addition to other parts. This is the most robust contact setting since all possible external contacts should be detected.

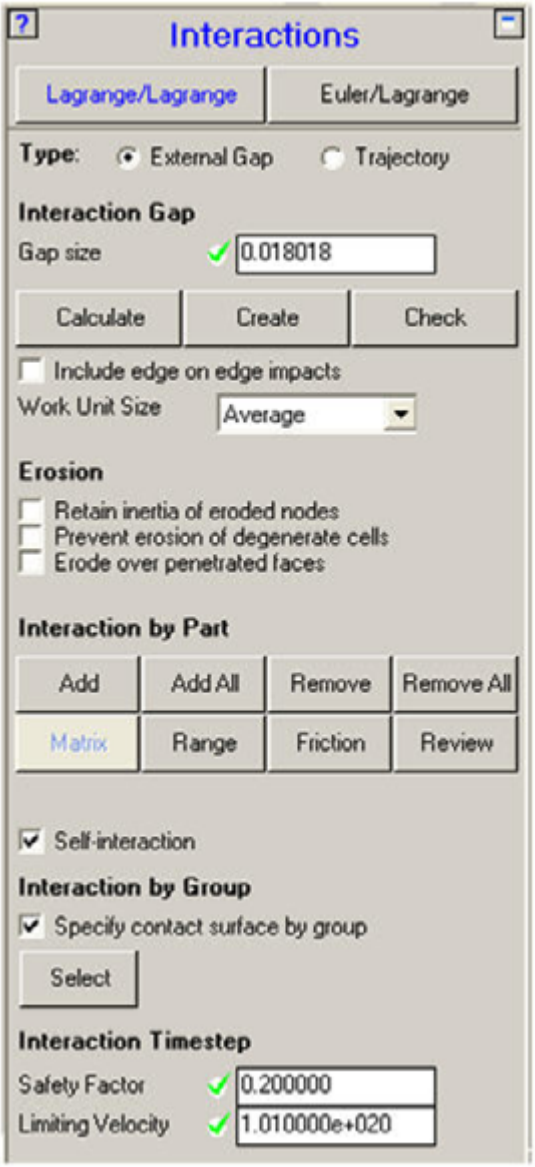
Self-interaction Tol

The self-interaction option enables automatic erosion when an element deforms such that one of its nodes comes within a specified distance of one of its faces. This option will prevent that volume elements become degenerated.

The specified distance is calculated using the Self-interaction Tol. value which is a factor in the range 0.1 to 05. This factor is multiplied by the smallest characteristic dimension of the elements in the mesh to give a physical dimension.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

14.1.10. Interactions - Lagrange/Lagrange - 3D Gap Contact Options



Interaction Gap (External Gap options)

The External Gap contact algorithm is implemented for all Lagrange, ALE, shell, beam and unstructured solvers.

A gap is required for the interaction logic. The size of this gap must be in the range 1/10 to 1/2 the dimension of the smallest element face of Parts involved in interactions.

There are two contact algorithms available in 3D, External Gap and Trajectory

For most models you should set the gap size close to the smallest allowed value (1/10 the dimension of the smallest element face).

Gap size

Enter a value for the Gap size in this field or click **Calculate** to have Autodyn set it for you.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Calculate

Click this button to have Autodyn automatically calculate the Gap size for you. The Gap size will be set to 1/10 the dimension of the smallest element face of Parts involved in interactions.

Create

All interacting Parts in your model must be separated by at least the Gap size. Click this button to automatically insert the required gap between Parts.

You may want to insert gaps manually using the **Translate** and / or **Scale** options in the zoning Panel Zoning panel.

Check

Click this button to check that the Gap size is valid and that all Parts involved in interactions are separated by the Gap size.

Include edge on edge impacts

Check this box if you want the interaction calculation to include testing for edge on edge impacts (significantly more computing is required if you check this box).

Work unit size

Select the size of work unit to be used for interactions:

Average

The work unit size is based on the average cell dimension.

Smallest

The work unit size is based on the smallest element dimension.

Largest

The work unit size is based on the largest element dimension.

Erosion

Retain inertia of eroded nodes

Check this box if you want to retain the inertial of eroded nodes (otherwise eroded nodes are removed from the model).

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Prevent erosion of degenerate elements

Check this box if you want the interaction calculation to be used to stop elements from becoming degenerate. This option only works for Lagrange volume elements.

> **Warning:**
>
> Activating this option can lead to unphysical results, particularly if a large Gap size is used.

Erode over penetrated faces

Check this box if you want elements with over penetrated faces to be eroded.

You should only check this box if you know this option is required to successfully complete your calculation.

Interaction by Part

Add

Sets interactions between Parts.

Add All

Sets all Parts to interact with each other.

Remove

Removes interactions between Parts.

Remove All

Removes all interactions between Parts.

Matrix

Sets interactions using a Part matrix.

Range

Sets the index range of each Part that will be checked for interactions.

Friction

Sets static friction coefficients between different Parts.

Review

Enables a review of all of your interaction settings.

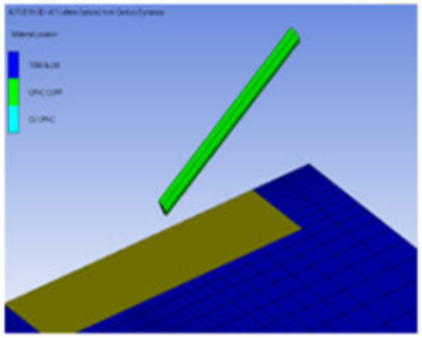
Self interaction of Parts

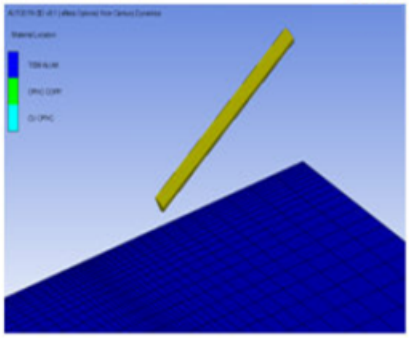
Check this box if you want Parts to interact with themselves (i.e. test for self-impact).

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

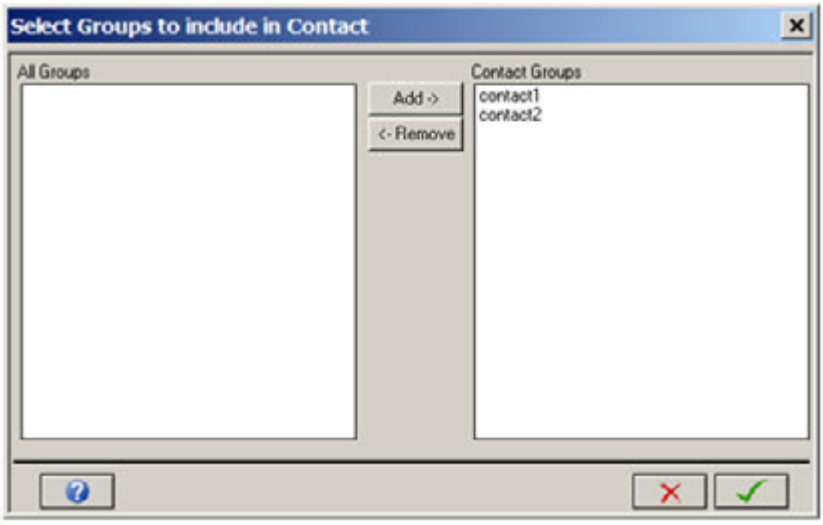
Interaction by Group

In many calculations, the contact region may be relatively small in comparison to the entire model, and as the interactions calculation is numerically intensive, it would be beneficial if the scope of its work were limited to this smaller area. The Group Contact option enables you to select a set of face groups to describe the extent of the interactions in unstructured models.



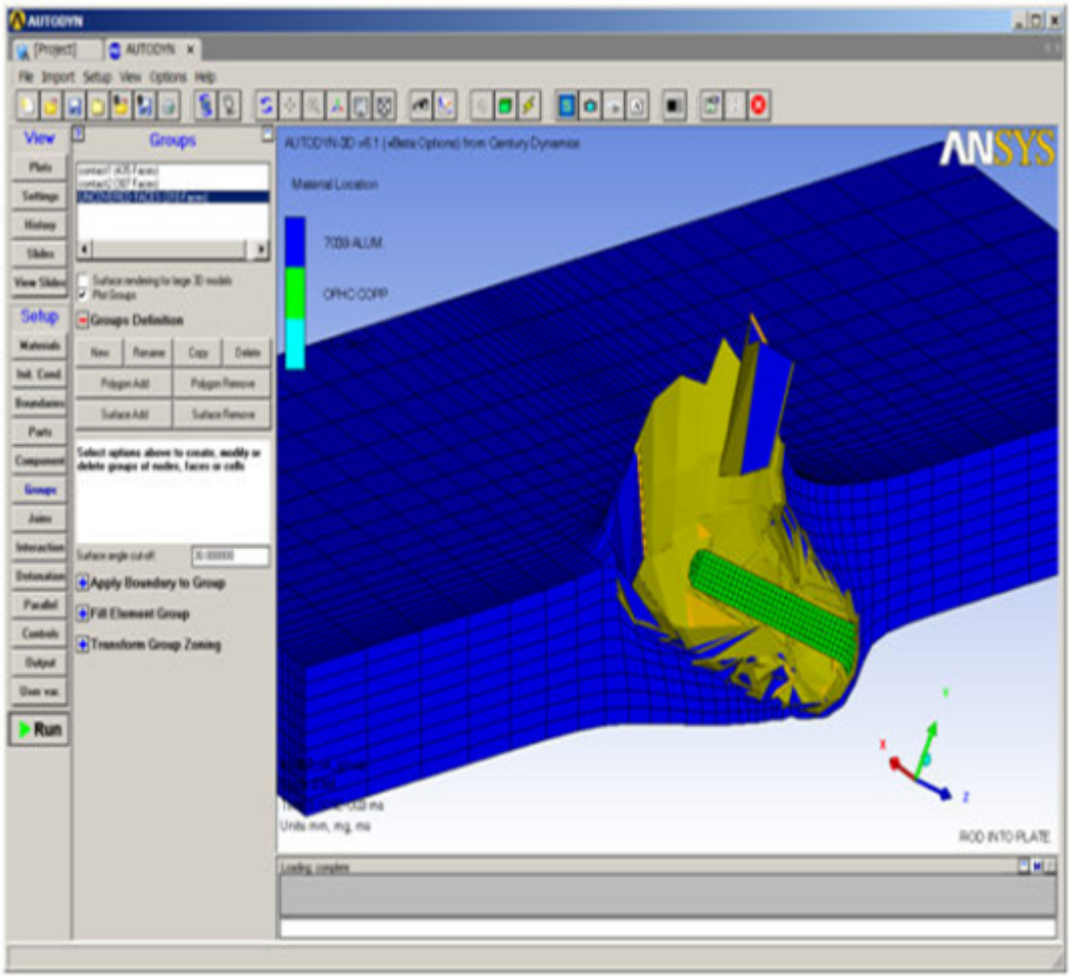


Select the "Specify Group Contacts by Group" toggle and click Select, to open the "Select Groups to include in Contact" dialog, and use the Add/Remove button to specify which face groups are to be included in the contact calculations.



During execution, any faces eroded will be removed from the group, and any new faces uncovered will be added to a group named "Uncovered faces". Unstructured Beams and SPH can also be included as a node group to participate in contact.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Note that Group Contact is additive to Part contact. Selecting a Part for contact results in all external faces in the Part being checked for contact. Group contact should be used to add additional external faces from Parts that have been excluded from contact in the Part interaction matrix.

Group contact is only available to faces of unstructured Parts and doesn't include friction.

Interaction Timestep

Autodyn automatically calculates an Interaction Timestep based on the Gap size and the maximum velocity of the Parts involved in interactions.

Safety Factor

The safety factor you want to use to calculate the Interaction Timestep.

You cannot enter a value greater than the default value (0.2).

Limiting Velocity

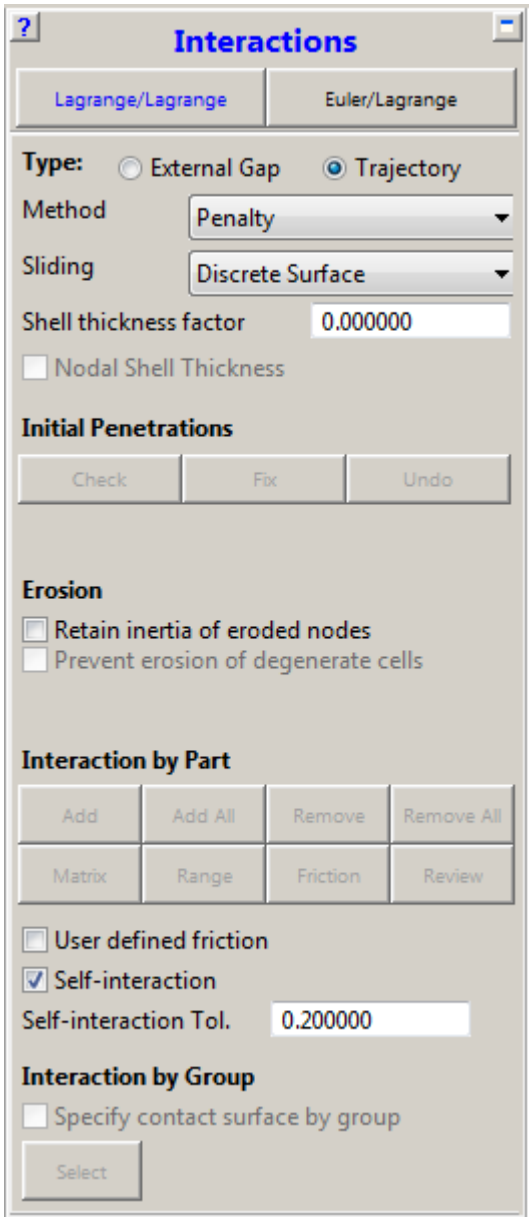
The maximum velocity you want to use for calculating the Interaction Timestep.

This option is useful if you have large velocities controlling the timestep that are not involved in interactions.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

The default limiting velocity is larger than any velocity your model should encounter.

14.1.11. Interactions - Lagrange/Lagrange - 3D Trajectory Contact Options



Interaction Trajectory (Trajectory options)

The Trajectory contact algorithm is implemented for all unstructured Lagrangian solvers and SPH.

Method

Method in which a contacting node is pushed back to the true contact position during the computational cycle.

There are three methods available:

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

None

No contact defined.

Penalty

If a contact event is detected, a local penalty force is calculated to push the node back to the face. Equal and opposite forces are calculated on the nodes of the face in order to conserve linear and angular momentum. The applied penalty force will push the node back towards the true contact position during the cycle. However, it will take several cycles to satisfy the contact condition. Kinetic energy is not necessarily conserved. The conservation of energy can be tracked using the energy time history.

Decomposition Response (DCR)

All contacts that take place at the same point in time are first detected. The response of the system to these contact events is then calculated to conserve momentum and energy. During this process, forces are calculated to ensure that the resulting position of the nodes and faces does not result in further penetration at that point in time. The decomposition response algorithm is more impulsive (in a given computational cycle) than the penalty method. This can give rise to large hourglass energies and energy errors.

Sliding

The algorithm used to determine how a node in contact slides along a surface. When a node that is in contact slides to the edge of a face on a mesh, the next face that the node slides on can be determined using two methods:

Discrete Surface

This is the default option. The next face that the contact node slides on is determined using the contact detection algorithm.

Connected Surface

The next face that the contact node slides on is determined using the mesh connectivity.

The Discrete Surface option will provide the most time efficient solution, but penetrations of nodes may be seen in situations where the faces that the nodes are sliding on are experiencing large deformations or rotations. When such penetrations occur, it is recommended the user switches to the Connected Surface option.

Shell thickness factor

The value of the factor must be between 0.0 and 1.0, and determines the amount of the shell thickness that is taken into account for the interaction distance. Typically, a value of 0.0 or 1.0 should be chosen.

Interaction in the solver is always taking place between a node and a face (contact surface). You can enable two (complementary) algorithms to take the shell thickness into account:

1. Shell thickness for the faces which will offset the faces

2. Shell thickness for the nodes which creates a "sphere" around the node

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

In order to use one or both of these thickness algorithms you can:

1. Set the Shell thickness factor to a value other than zero to activate the Shell thickness algorithm for the faces
2. Enable Nodal Shell Thickness to activate the Shell thickness algorithm for the nodes, in addition to Shell thickness for the faces

Interaction Behavior with Shell Thickness

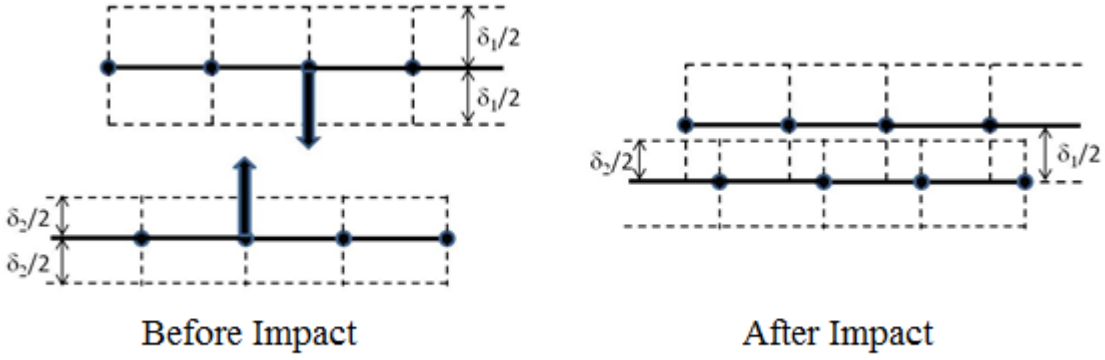
Setting the factor to a value other than zero means that the contact surface is positioned at (0.5 x shell thickness x factor) on both sides of the shell mid plane.

A factor of 0.0 means that the shell has no contact thickness and the contact surface is positioned at the shell mid plane. Note that with this setting the nodal shell thickness can not be activated separately.

The contact area of a node depends on the setting for Nodal Shell Thickness. If it is disabled, the node is always located at the mid-surface of the shell (Situation I (p. 425)). If it is enabled the node is located at a spherical distance of half the thickness away from the physical node location (Situation II (p. 425)).

Situation I

Two shell parts with thickness δ_1 and δ_2 will not contact at a distance of $(\delta_1/2 + \delta_2/2)$, but at a distance which is half of the largest shell thickness as is depicted below.

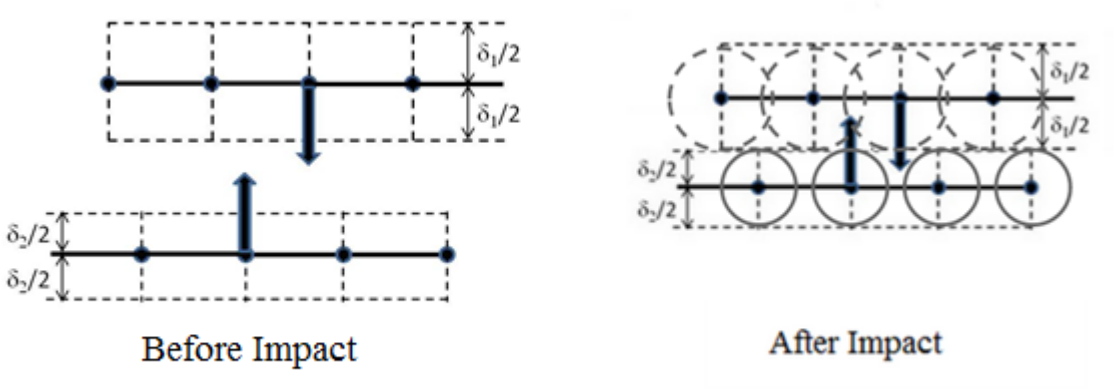


Note that for shell node on solid face impacts, the node will be able to get to within zero distance of the solid face; the thickness for the shell nodes will not be taken into account. The solid nodes will, however, find the shell faces at contact distance.

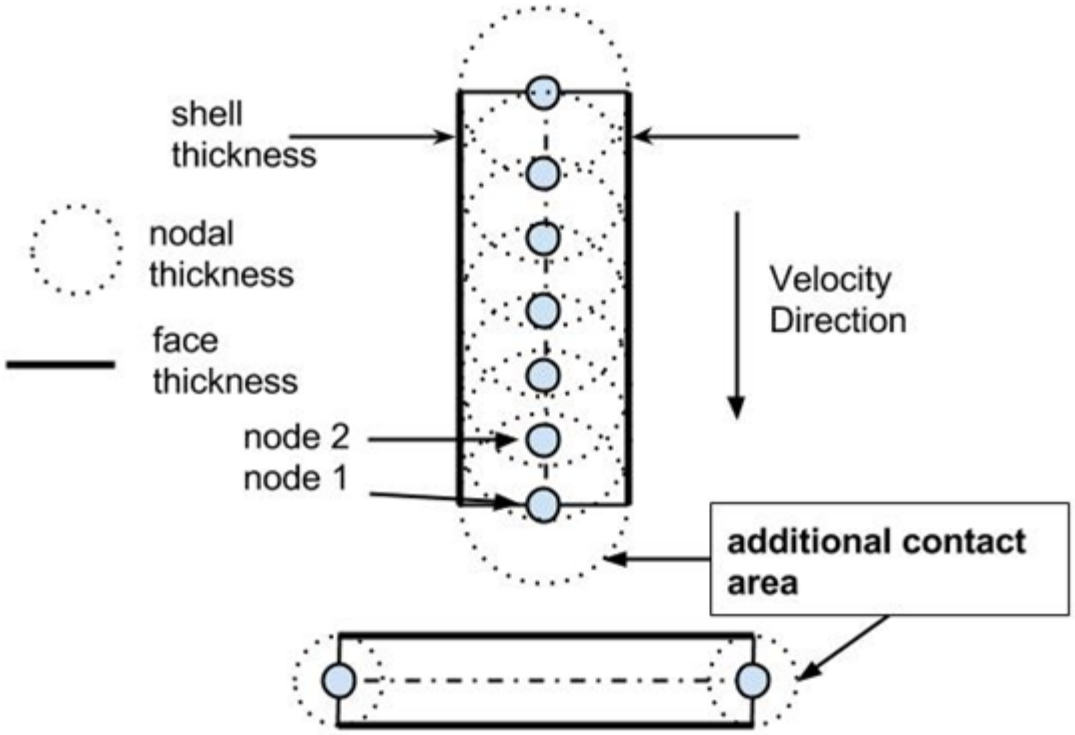
Situation II

By enabling the nodal shell thickness, two shell parts will contact at a distance of $(\delta_1/2 + \delta_2/2)$. From a physical point of view this is correct, as can be seen in the picture below.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Note that care should be taken for nodes that are on or close to a free edge of the shell surface because the node may find contact in an unexpected manner due to the spherical contact around these nodes. This is shown below in a 2D manner, where for example node 1 and 2 have an additional contact area which extends beyond the geometry.



Initial Penetrations

Check

On clicking this button, Autodyn will search the model for initial penetrations of nodes into surfaces. If any penetrating nodes/surfaces are found, a new Group will be created so that

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

they can be identified/displayed using the Groups panel. Nodes that initially penetrate surfaces will be "missed" by the contact algorithm.

Fix

When this option is selected the initially penetrated nodes are set back to the closest contact surface and no initial penetrations are present at the start of the analysis.

This option is meant for situations where small initial penetrations occur (for example because of round-off) and should be used with care because the analysis model geometry is changed.

Note that it may not be possible to remove all initial penetrations with this algorithm. Features are available in the DesignModeler and Meshing application to prevent initial penetrations occurring.

Undo

This option will restore the geometry of the analysis model before the initial penetrations were fixed.

It is recommended that the fixed initial penetrations are checked before the model is modified further otherwise it may not be possible to undo the last fix operation.

Erosion

Retain inertia of eroded nodes

Check this box if you want to retain the inertial of eroded nodes (otherwise eroded nodes are removed from the model).

Interaction by Part

Add

Sets interactions between Parts.

Add All

Sets all Parts to interact with each other.

Remove

Removes interactions between Parts.

Remove All

Removes all interactions between Parts.

Matrix

Sets interactions using a Part matrix.

Range

Sets the index range of each Part that will be checked for interactions.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Friction

Sets static friction coefficients between different Parts.

Review

Enables a review of all of your interaction settings.

User defined friction

Aside from the friction definition through the interaction by Part method described above there is also a user defined friction option available through the use of a user-subroutine called EXFRICTION.

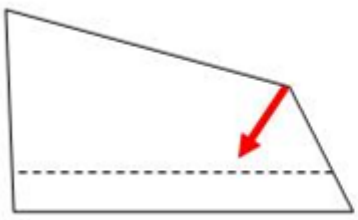
Self Interaction

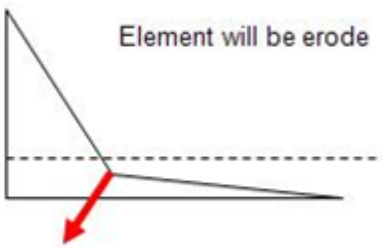
If the self-interaction option is switched on the contact detection algorithm will also check for external nodes of a part contacting with faces of the same part in addition to other parts. This is the most robust contact setting since all possible external contacts should be detected.

Self-interaction Tol.

The self-interaction option enables automatic erosion when an element deforms such that one of its nodes comes within a specified distance of one of its faces. This option will prevent volume elements becoming degenerate.

The specified distance is calculated using the Self-interaction Tol. value which is a factor in the range 0.1 to 0.5. This factor is multiplied by the smallest characteristic dimension of the elements in the mesh to give a physical dimension.

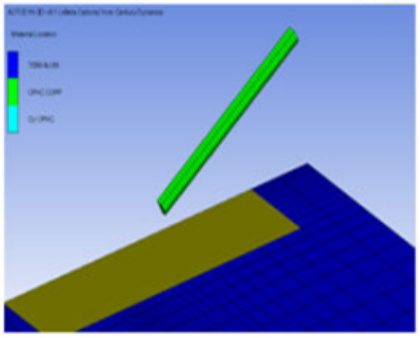


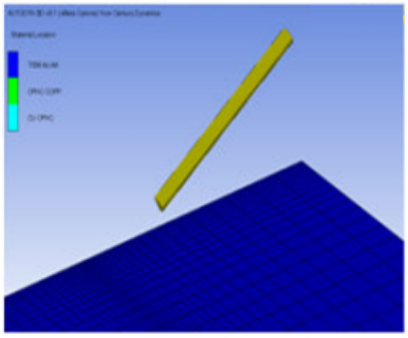


Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

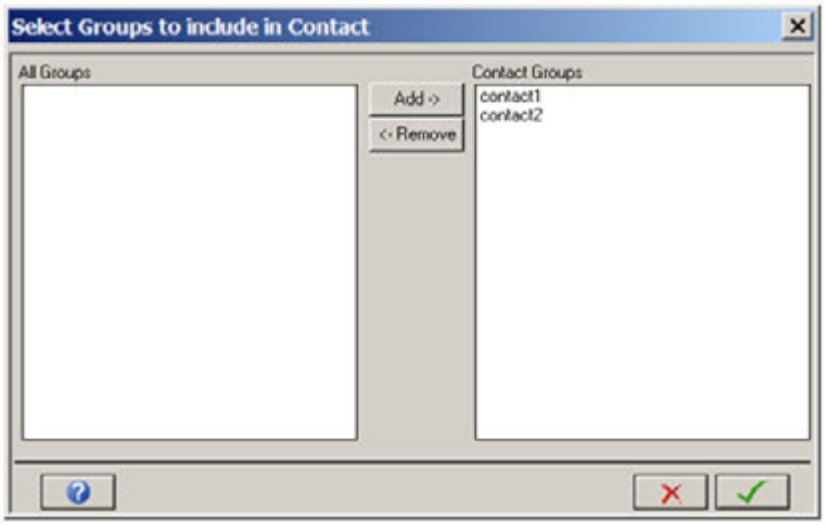
Interaction by Group

In many calculations, the contact region may be relatively small in comparison to the entire model, and as the interactions calculation is numerically intensive, it would be beneficial if the scope of its work were limited to this smaller area. The Group Contact option enables you to select a set of face groups to describe the extent of the interactions in unstructured models.



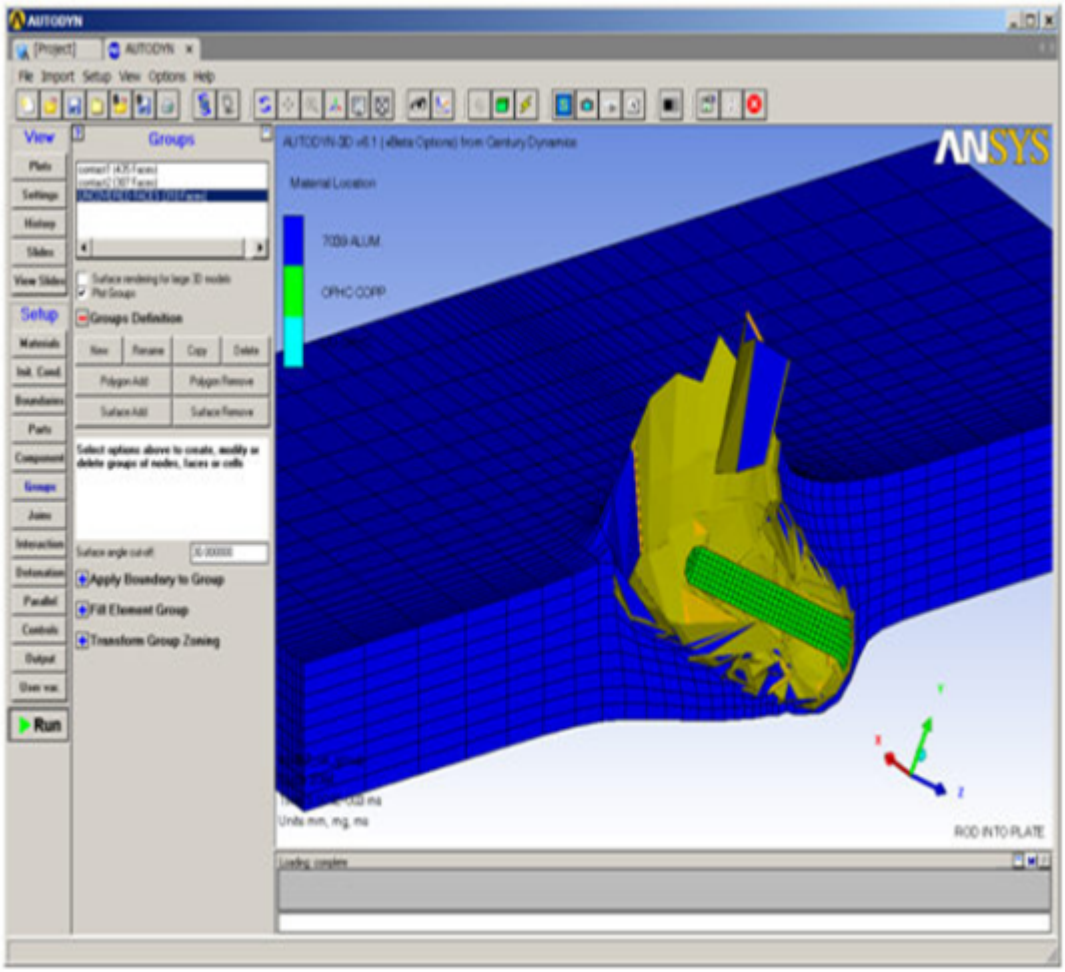


Select the "Specify Group Contacts by Group" toggle and click Select, to open the "Select Groups to include in Contact" dialog, and use the Add/Remove button to specify which face groups are to be included in the contact calculations.



During execution, any faces eroded will be removed from the group, and any new faces uncovered will be added to a group named "Uncovered faces". Unstructured Beams and SPH can also be included as a node group to participate in contact.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



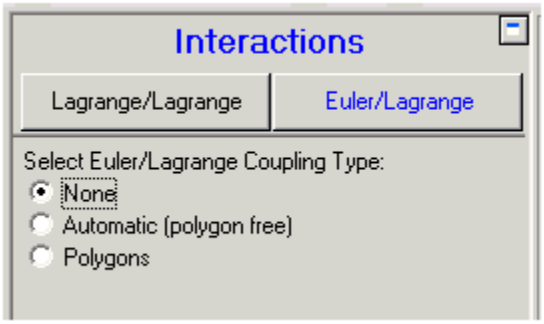
Note:

Note that Group Contact is additive to Part contact. Selecting a Part for contact results in all external faces in that Part being checked for contact. Group contact should be used to add additional external faces from Parts that have been excluded from contact in the Part interaction matrix.

Friction cannot be specified on a group-by-group basis. However, friction coefficients set on a part-by-part basis as described in the section on Interaction by Part (p. 427) will apply regardless of whether any interactions by part have been set.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

14.2. Interactions - Euler/Lagrange (2D)



This panel lets you set up interactions (coupling) between 2D Lagrange Parts and Euler Parts.

Select Euler/Lagrange Coupling Type

Select the type of coupling you want to muse for Euler/Lagrange interactions.

None

Do not compute any Euler/Lagrange coupling.

Automatic

Let Autodyn set up coupling automatically.

Polygons

Set up interactions manually using Polygons.

Automatic coupling

Recommended if you want all Lagrange Parts to interact with all Euler Parts and you have no Shell Parts.

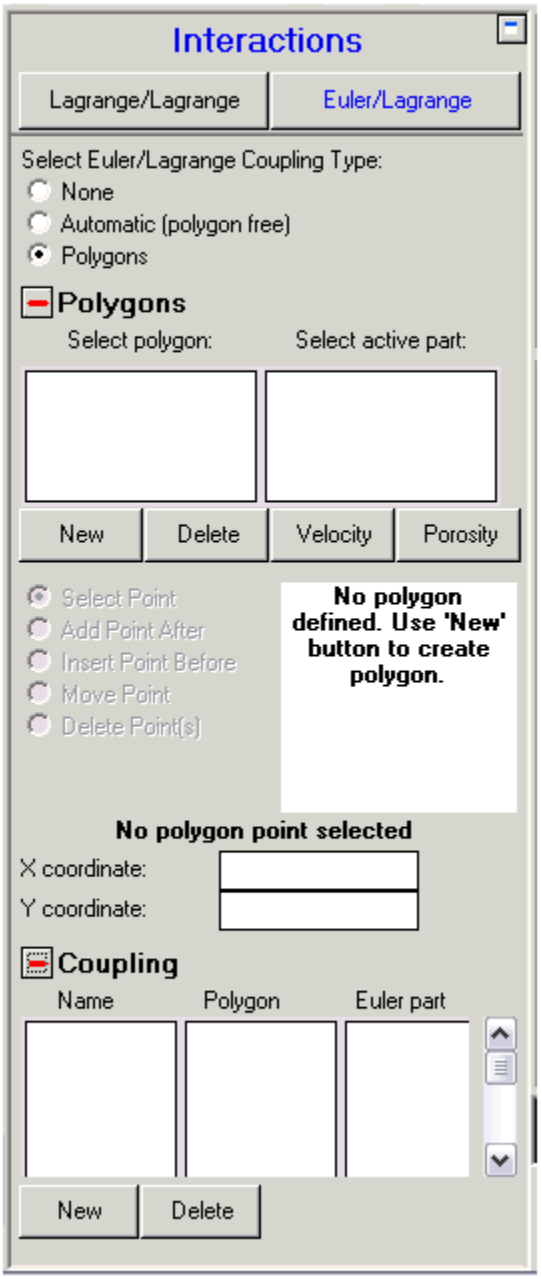
If you want more control in defining which Parts should be coupled or have Shell Parts that you want to couple to Euler, use **Polygon coupling**.

Erosion

Can be used with **Automatic coupling**, but not with **Polygon coupling**.

If you select **Polygon coupling**, further input dialog is presented :

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



To set up an Euler/Lagrange interaction using Polygons, you must first generate one or more Polygons and attach the polygon points to nodes on the surface of the Lagrange Parts that you want to interact with Euler.

Then you must specify which Polygons you want to couple with which Euler Parts.

This dialog lets you generate your Polygons and set up coupling between your Polygons and the Euler Parts in your model.

Polygons

Select Polygon

Lists all the Polygons you have created. You can select a Polygon in this list.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Select active part

Lists all the Parts that can be coupled to Polygons (that is, all Lagrange, ALE and Shell Parts). You can select a Part in this list.

New

Creates a new Polygon.

Delete

Deletes the selected Polygon

Velocity

Applies a velocity to the selected node.

Porosity

Sets porosity for the selected Polygon.

Add First Point (only visible when no points are defined)

Generates the first point for the selected Polygon (use **Shift**+right mouse button to place the point).

Select Point

Selects a point of the selected Polygon (use **Shift**+right mouse button to select the point).

Add Point After

Adds a point after the selected polygon point (use **Shift**+right mouse button to place the point).

If you place the added point close to a node on the surface of the selected active Part, the point will automatically be attached to that node.

If you immediately follow this by placing a second point close to another surface node of the same selected active Part, multiple points will be generated attached to all surface nodes between the two placed nodes in an anti-clockwise direction.

Insert Point Before

Inserts a point before the selected polygon point (use **Shift**+right mouse button to place the point)

If you place the inserted point close to a node on the surface of the selected active Part, the point will automatically be attached to that node.

If you immediately follow this by placing a second point close to another surface node of the same selected active Part, multiple points will be generated attached to all surface nodes between the two placed nodes in an anti-clockwise direction.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Move Point

Moves the selected polygon point to a new location (use **Shift**+right mouse button to place the point at its new location)

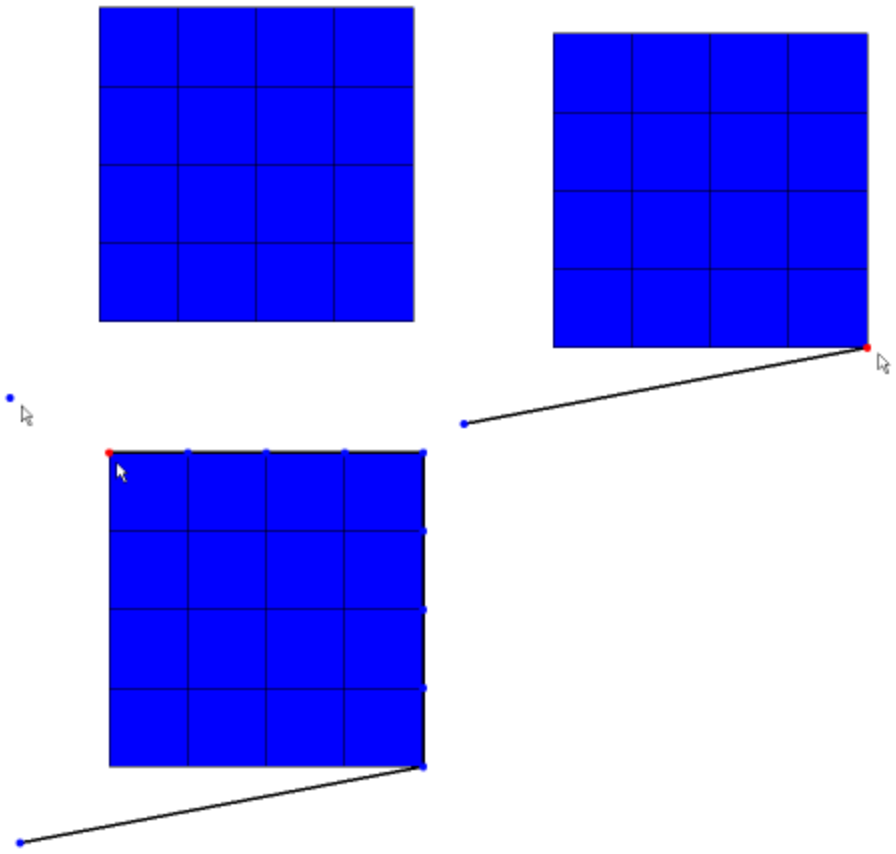
Delete Point(s)

Deletes one or more polygon points. The first point to be deleted is the selected polygon point. Use **Shift**+right mouse button to select the last point to be deleted. All points from the selected point to last point inclusive will be deleted.

X/Y Coordinates

These fields display the coordinates of the selected polygon point. You can change the location of this point by entering new values in these fields.

Example 14.1: Adding Points



- Add first point (Single point not attached to grid).
- Add point after (Single point attached to grid).
- Add point after (Multiple points generated anti-clockwise around grid).

The selected polygon node is always shown in red.

The last point of Polygon is automatically connected to first point to close the Polygon.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Coupling, Name / Polygon / Euler Part

These boxes list the interaction sets you have defined for Polygon coupling.

Name

The name of the interaction set.

Polygon

The Polygon that is coupled to an Euler Part.

Euler Part

The Euler Part that the Polygon is coupled to.

You can select a set in this list.

New

Adds a new interaction set.

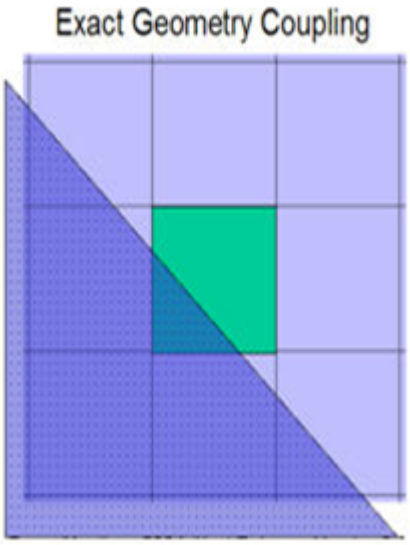
Delete

Deletes the selected interaction set.

14.2.1. Interactions - Euler/Lagrange - Introduction

The Euler/Lagrange interaction (Fluid-Structure Interaction) in Autodyn couples the Eulerian domain to the Lagrangian structural domain.

This coupling algorithm in Autodyn finds the intersection between the external faces of the interacting structures and the cells and faces of the Euler grids. These intersection calculations give an exact calculation (within numerical precision) of the covered fraction of the Euler cells and faces, as shown in the left hand figure below. These properties are used in the 3D Euler solvers for coupled simulations.

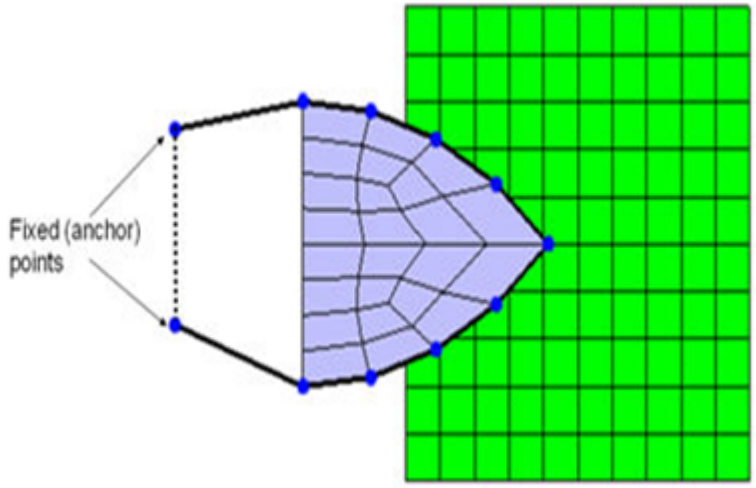


The coupling can be chosen to be fully automatic (2D/3D) or by means of polygons (2D).

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Polygon coupling

This coupling technique is available in 2D to set up an Euler/Lagrange interaction where one or more Polygons are defined and the polygon points are manually attached to nodes on the surface of the Lagrange Parts that interact with Euler. All structured Lagrangian element types can be coupled to Euler using polygons. Unstructured parts cannot be coupled to Euler using polygons; in that case the automatic coupling needs to be used.

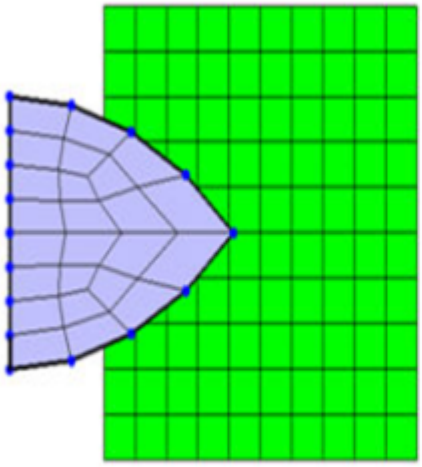


Automatic coupling

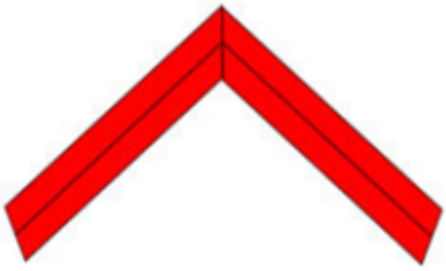
This coupling technique is available in 2D as well as in 3D. Autodyn will automatically set up the Euler/Lagrange interactions and the only thing you have to define is which parts interact with Euler. Automatically all outside surface segments of all selected Lagrangian parts are generated and coupled to Euler.

In 2D the automatic coupling is only available for structured and unstructured Lagrangian volume elements.

In 3D the automatic coupling is available for structured and unstructured Lagrangian volume and shell elements. For shell elements an artificial thickening of the shell parts is used.



Automatic generation of outside surface



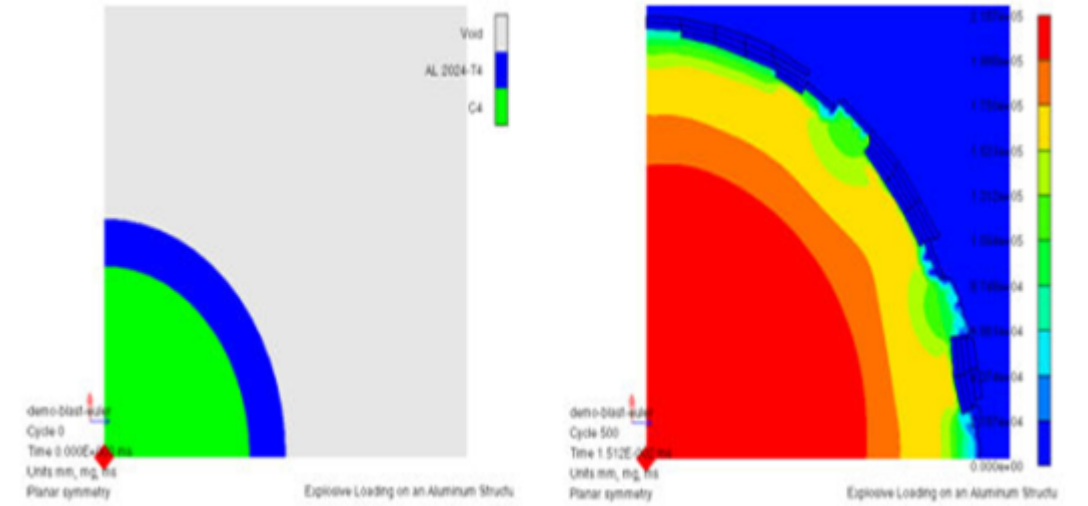
Artificial shell thickening for coupling

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Coupling and Erosion

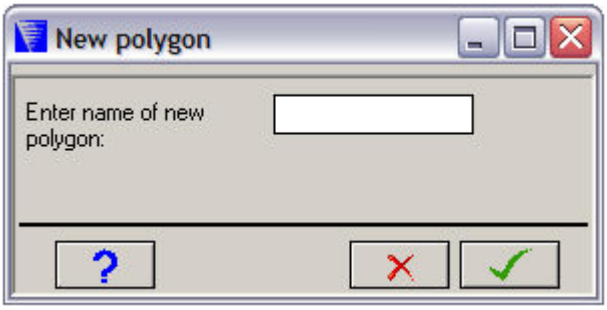
Erosion in combination with Euler/Lagrange coupling can be used for automatic coupling in 2D as well as 3D.

When elements that are connected to the coupling surface are eroded the coupling surface automatically will be updated for the new uncovered Lagrangian element faces.



Example simulation of fragmenting ring using Euler-Lagrange Coupling and requiring automated update of coupling surfaces during the simulation

14.2.2. Interactions - Euler/Lagrange (2D) - New Polygon



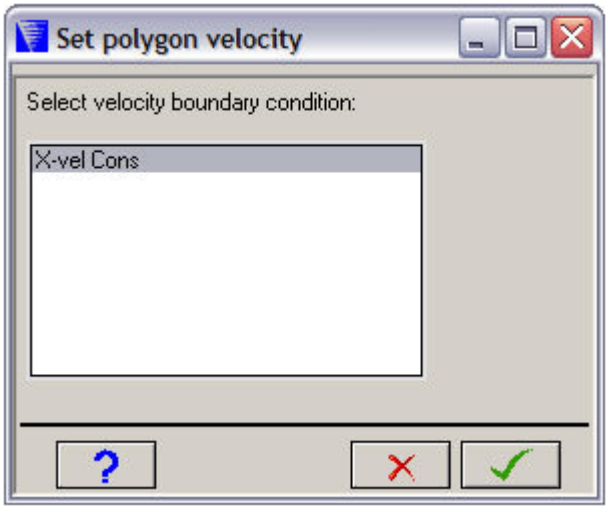
This window lets you create a new Polygon for Euler/Lagrange coupling.

Enter the name you want to use for your new polygon in the field provided and click .

Once you have created your Polygon, you can add points and attached these points to Lagrange nodes using options in the Dialog panel.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

14.2.3. Interactions - Euler/Lagrange (2D) - Velocity

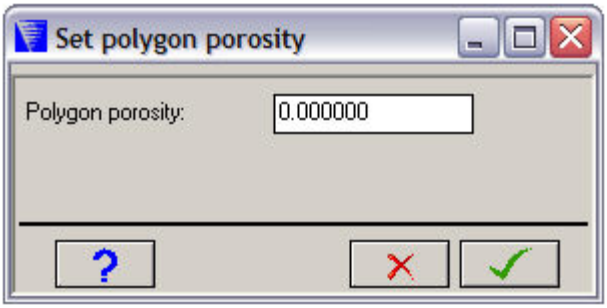


This window lets you attach a velocity boundary condition the selected polygon.

All polygon points not attached to a Part will use this velocity.

Select the boundary condition you want to use from the list and click .

14.2.4. Interactions - Euler/Lagrange (2D) - Porosity



This window lets you set porosity for your Polygon.

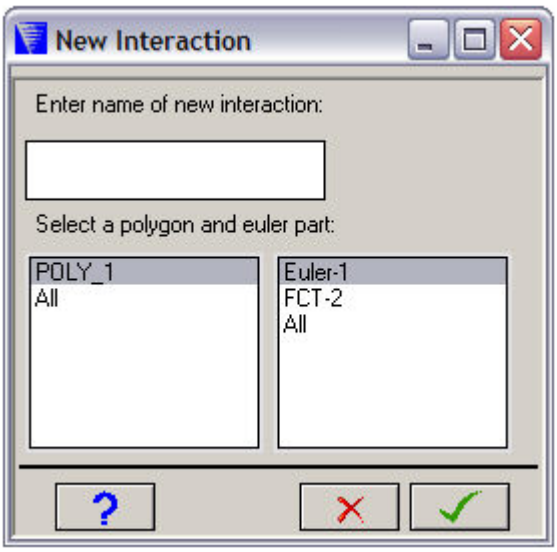
By default, a Polygon completely covers the Euler Parts it is coupled to.

Setting porosity greater than zero allows partial flow in the Euler Parts under the Polygon.

Enter a porosity factor between 0.0 and 1.0.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

14.2.5. Interactions - Euler/Lagrange (2D) - New Interaction



This window lets you define a new interaction set for Polygon coupling.

Enter name of new interaction

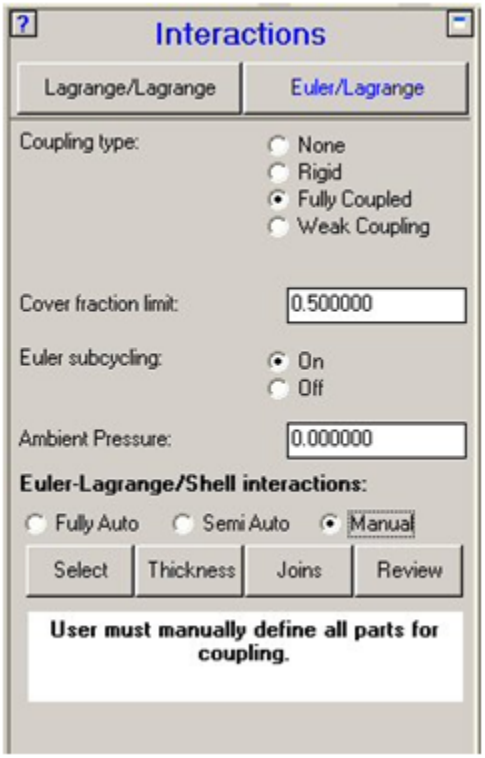
The name you want to use for the new interaction set.

Select a Polygon and Euler Part

Select the Polygon in the left box that you want to be coupled to the Euler Part you select in the right box.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

14.3. Interactions - Euler/Lagrange (3D)



This panel lets you set up interactions (coupling) between 3D Lagrange Parts and Euler Parts.

Coupling Type

Select the coupling type you want to use.

None

No coupling is defined.

Rigid

Fill Parts are used to define rigid boundaries for Euler.

Fully Coupled

Lagrange Parts interact with Euler Parts dynamically.

Weak Coupling

A fast method for having Lagrange and Euler Parts interact dynamically.

Cover fraction limit

The cover fraction limit. This is used to determine when a partially covered Euler cell is blended to a neighbor.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Euler subcycling

Select whether you want Euler subcycling **On** or **Off**

Euler-Lagrange/Shell interactions

Fully Auto

All structural parts will automatically be coupled with all Eulerian parts in the model. Shell parts will be thickened automatically to fit the initial Euler region they reside in.

Review

Displays your Euler-Lagrange/Shell interactions.

Semi Auto

You must manually select all structural parts that will be coupled with all Eulerian parts in the model. Shell parts will be thickened automatically to fit the initial Euler region they reside in.

Select

Selects the structural Parts you want to couple to Euler.

Review

Displays your Euler-Lagrange/Shell interactions.

Manual

You must manually select all structural parts that will be coupled with all Eulerian parts in the model and also define the artificial thickness of each shell part in the interaction.

Select

Selects the structural Parts you want to couple to Euler.

Thickness

Specifies the shell thickness you want to use for the coupling calculation.

Joins

Specifies which shells are to be joined for the coupling calculation.

Review

Displays your Euler-Lagrange/Shell interactions.

Weak coupling settings

Select the weak coupling settings.

Browse

Enables you to select the location and name of the weak coupling field files to be used.

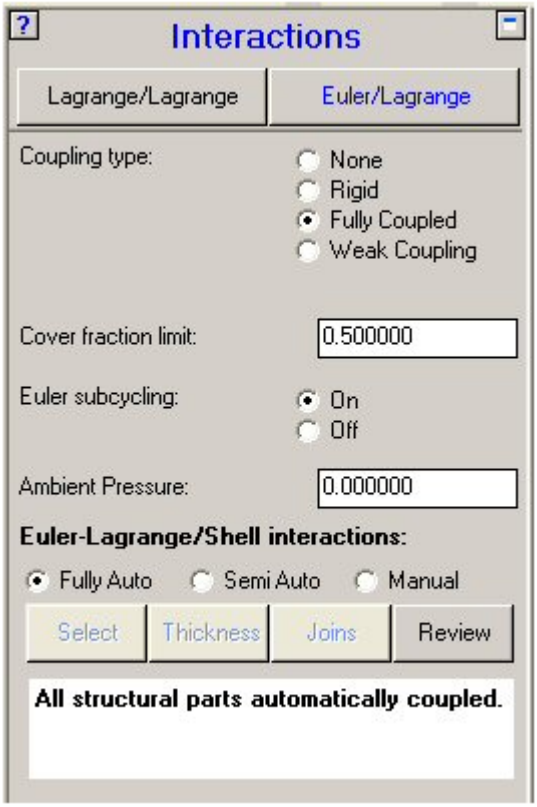
Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Geometry for mapping

Selection of the Lagrange geometry that is used for the remapping. You can select the initial geometry or the current geometry.

14.3.1. Interactions - Euler/Lagrange (3D) - Type

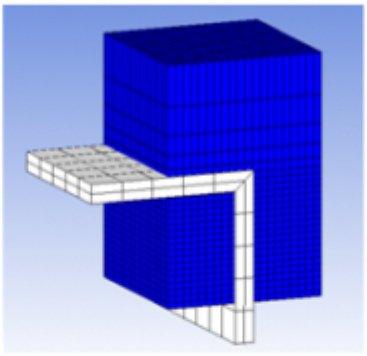
Select the type of automation you want to use for the Euler-Lagrange/Shell interaction definition.

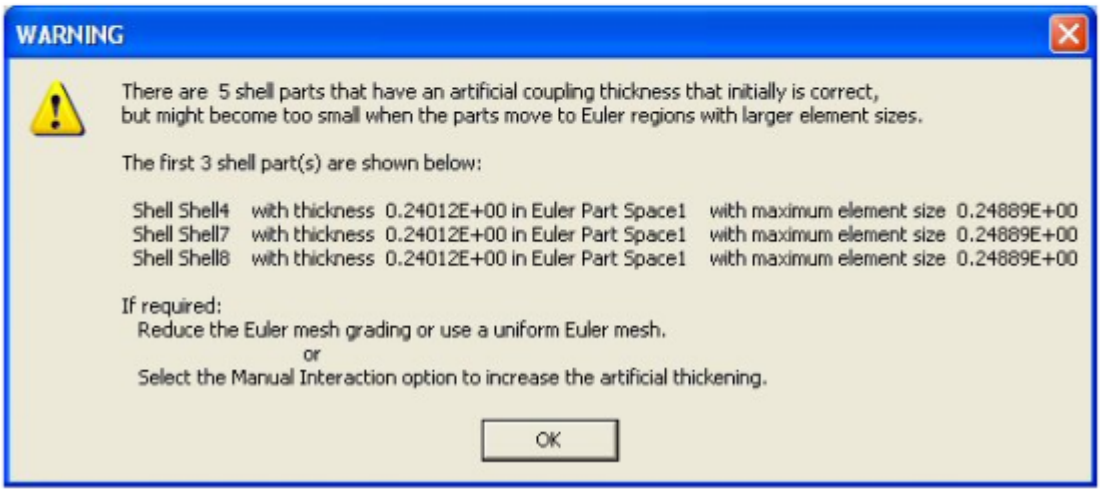


Fully Auto

The **Fully Auto** option will fully automate the set up of the Euler-Lagrange/Shell coupling. All structural Parts will be coupled to the Eulerian Parts in the model, and all shell Parts in the model will be thickened in such a way that they will fit the region of the Euler mesh they initial reside in. For each shell Part, the fully automatic method will compute the largest element diagonal of all Euler elements the shell part initially resides in. The artificial coupling thickness is then taken as 1.5 times the value of the largest element diagonal and is applied as a centered thickening. This will guarantee that thickened shell parts at the beginning of the analysis at least fully cover the Euler elements the shell resides in, which is a prerequisite for the coupling algorithm to function correctly. In cases where a large mesh grading is applied, the initial shell thickness might not be large enough when the shell Part moves into an Euler region with larger size elements and the Euler elements are not fully covered anymore. The automatic coupling method will notify you at the beginning of the analysis if such a risk exists and you can then decide to use a smaller grading of the Euler mesh if required. This typical warning message is shown below:

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

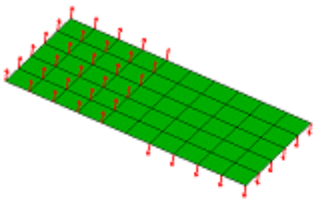
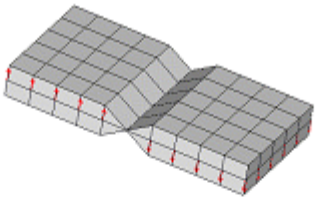
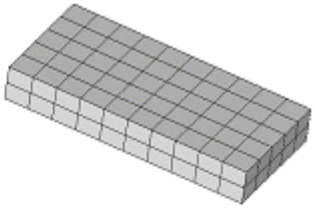


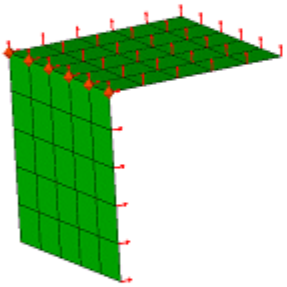
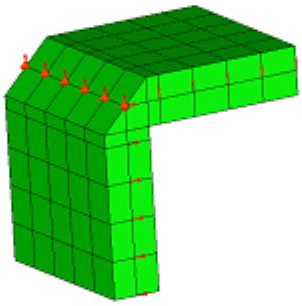
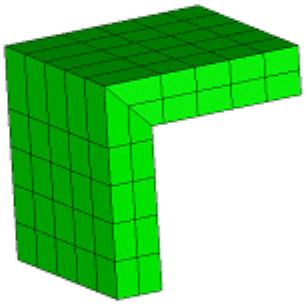


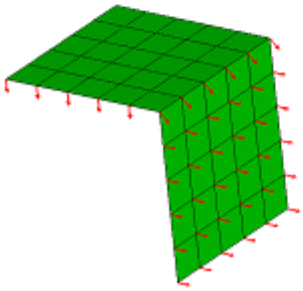
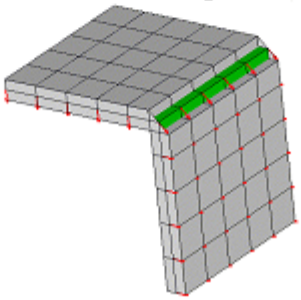
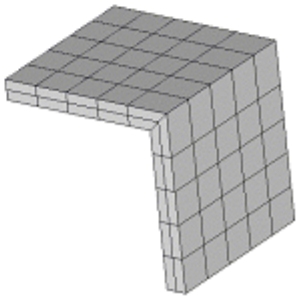
Aside from Part selection and automatic shell thickening, the fully automatic option also checks the orientation of the shell normals within a single shell part, and also the orientation of the normals of different shell parts at a (T)-joint. Based upon this information, the automatic method will create a consistent closed-coupling surface in a way similar to that you must do in a manual manner in cases where inconsistent normal directions would cause problems. The manual options available are reversal of normals under the Group option and the removal of Parts in the join normal calculation.

Some typical examples:

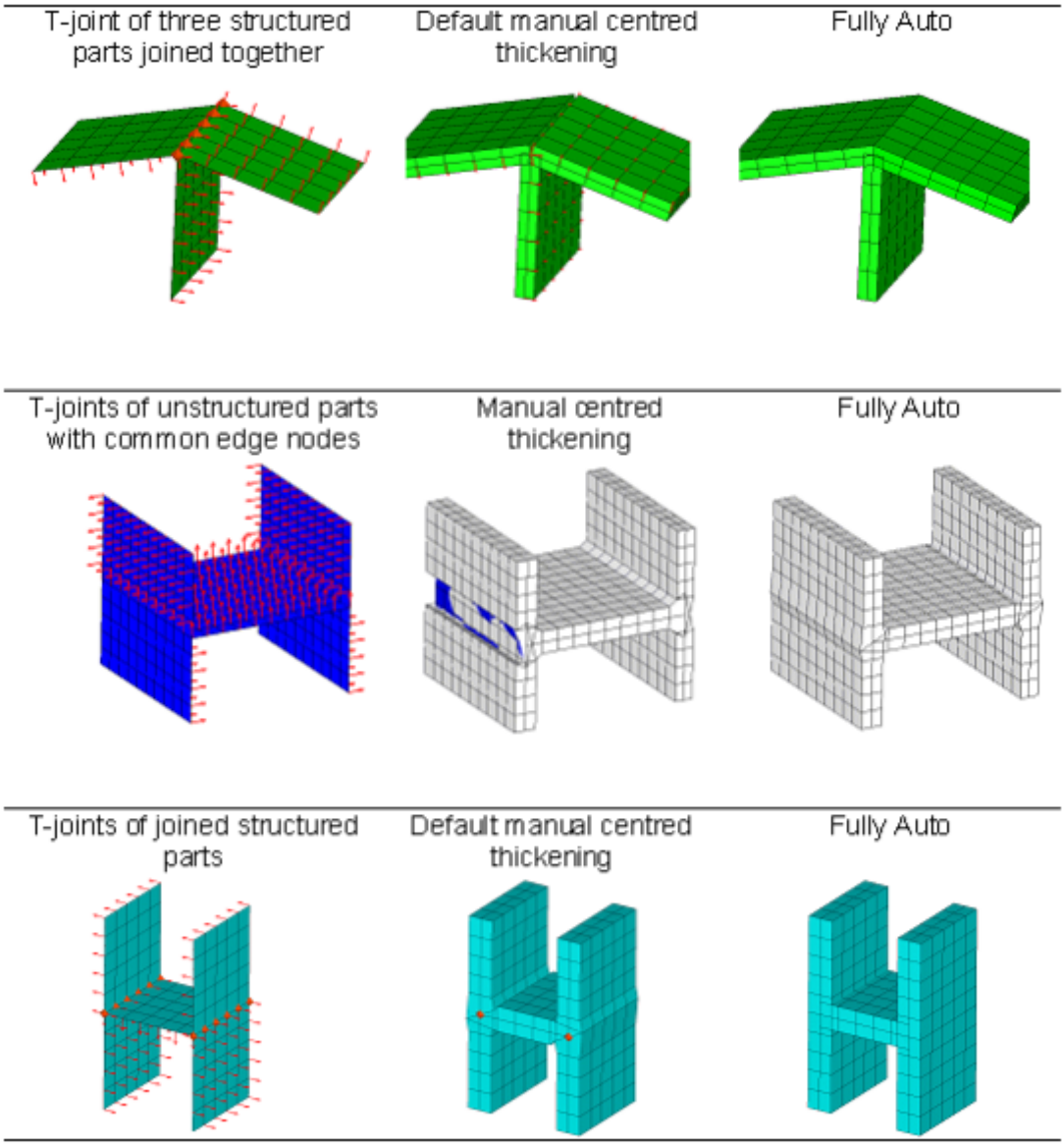
Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information
of ANSYS, Inc. and its subsidiaries and affiliates.

Single unstructured part	Default manual centred thickening	Fully Auto
		

Two structured parts joined together	Default manual centred thickening	Fully Auto
		

Two unstructured parts with common edge nodes	Default manual centred thickening	Fully Auto
		

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Semi Auto

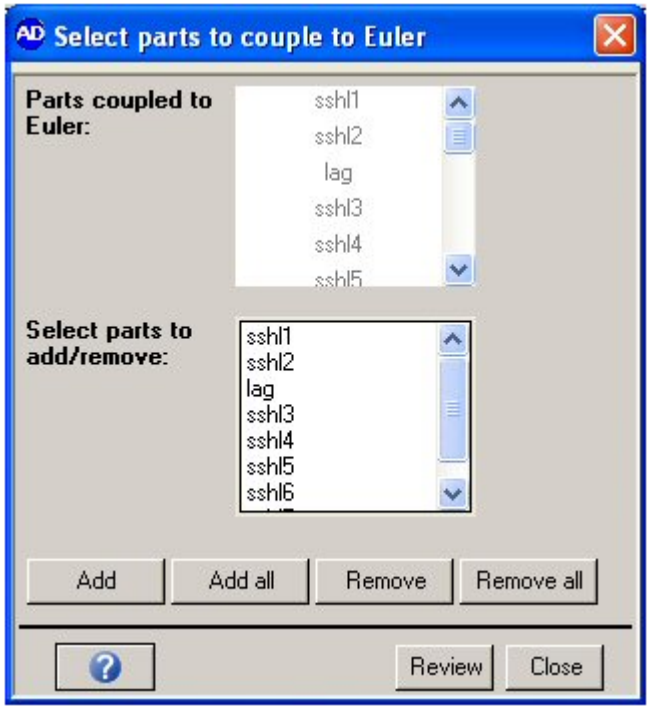
Semi Auto is similar to the Fully Auto option. The only difference is that for the Semi Auto option you must manually select the structural parts for interaction with the Euler Parts. The artificial coupling thickness of the selected shell Parts again is computed fully automatically.

Manual

When **Manual** is selected, you must define the complete coupling manually by selecting the structural Parts for Interaction, defining the artificial shell coupling thickness, and verifying that the coupling surface is a consistent closed surface.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

14.3.2. Interactions - Euler/Lagrange (3D) - Select



This window lets you select the Lagrangian Parts you want to couple to Euler.

Parts coupled to Euler

Shows you the current Parts that are coupled to Euler.

Select parts to add/remove

Shows all the Parts that can be coupled to Euler. You can select one or more Parts in this list.

Add

Adds the selected Parts to be coupled with Euler.

Add all

Adds all the Parts to be coupled with Euler.

Remove

Removes the selected Parts from being coupled with Euler.

Remove all

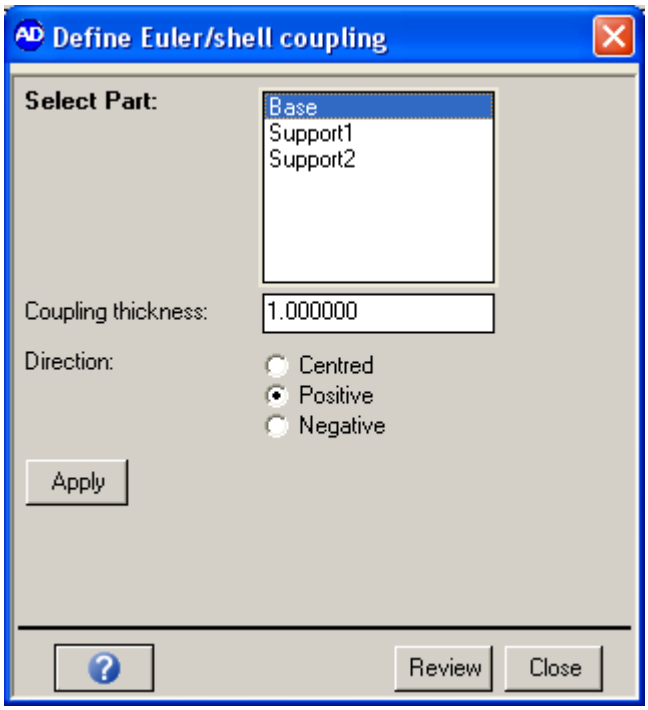
Removes all Parts from being coupled with Euler.

Review

Enables a review of all coupling with Euler.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

14.3.3. Interactions - Euler/Lagrange (3D) - Thickness



As well as being able to couple volume element solvers, it is also possible in Autodyn to couple shell solvers. For a shell solver, the parts do not have any geometric through thickness dimension, and as such cannot cover any volume in the Euler mesh. Therefore, you need to artificially thicken each shell part.

Select Part

This list contains all the shell parts in the model that are selected for coupling with Euler. If you select a part, the coupling thickness and direction of the thickening for that part is shown below.

Coupling Thickness

This shows the coupling thickness for the selected part. To modify the coupling thickness, enter the new value, and click the Apply button. For the coupling methodology to function correctly, the artificial thickness of a shell must be at least twice the dimension of the largest cell size in the surrounding Euler grid.

Direction

This shows the direction the shell is thickened in for the selected part. To modify the thickening direction, select the desired direction, and click the Apply button.

The coupling thickness can be applied on both sides of the midplane of the shell (Centered) or applied to one side of the shell, where Positive is in the direction of the normal of the shell.

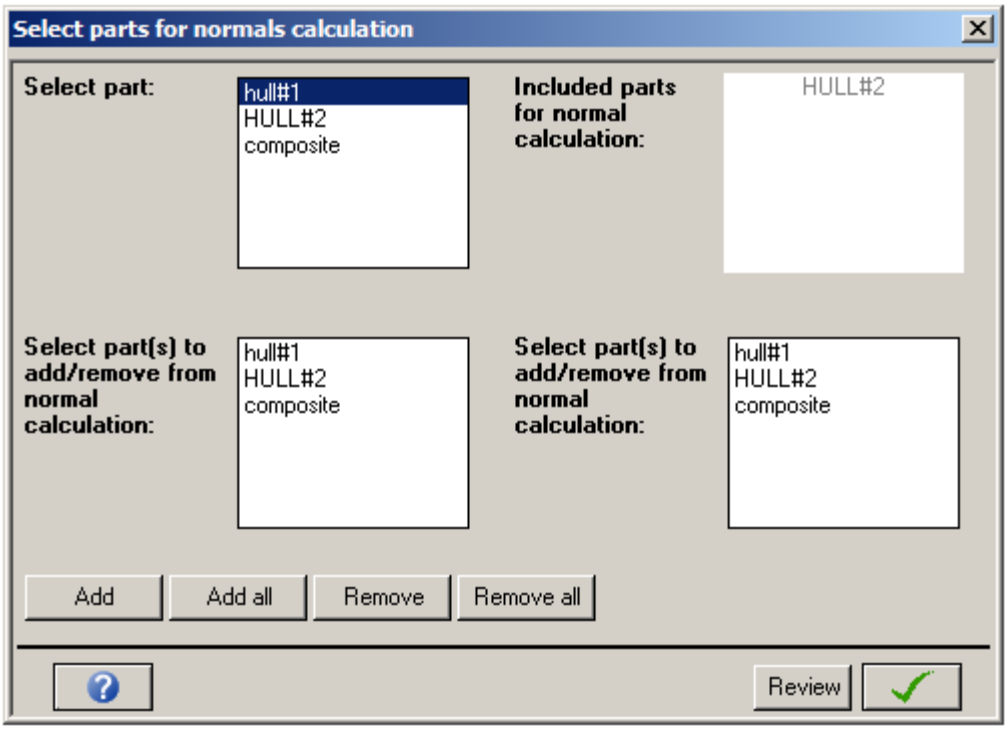
Review

If you select the review button, an HTML summary is opened that displays which shell parts have been included in the Euler coupling, the thickness and direction for these shell parts, and also the thickened shell joined matrix.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

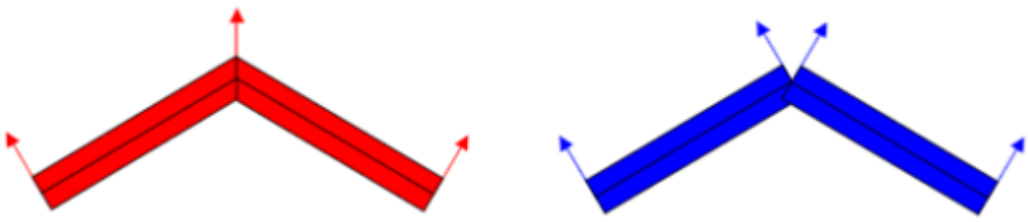
You can also view the thickened shells by switching on "Shell thickness" in the Plots panel, and choosing "Artificial Shell Coupling Thickness" in the Shell Thickness Settings panel.

14.3.4. Interactions - Euler/Lagrange (3D) - Joins



To ensure that volumes of joined thickened shells do not overlap as the shells deform, the thickening algorithm needs some information on which shell parts should be considered joined when computing the outward normals for shell nodes in the thickening geometry calculations. By default all thickened shell parts are set as joined for the coupling calculation. If this is not appropriate, for example for a T-junction between two shell parts, you should remove the joins for the coupling calculation. It is important to note that these join settings are only used in the coupling calculation and do not affect the structural joins.

This is illustrated in the figure below. On the left, the two shell parts are joined for thickening calculations, and the resulting thickened shells define a closed volume. On the right, the two shell parts are not joined for thickening calculations, and the shell overlap is not satisfactory for coupling.



Select Part

Shows all the shell parts that are coupled in Euler. When a part is selected in this list, all parts joined to it are displayed in the "Included parts for normal calculation" list.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Included parts for normal calculation

Shows all shell parts that are joined to the shell part selected in "Select Part"

Select part(s) to add/remove from normal calculation

These two fields are used to add or remove parts from the normal calculation. For example, a model has shell parts "hull#1", "hull#2" and "composite", as shown in the image of the window at the top of the page. To include the normals between the first two parts, select "hull#1" in the left hand list, "hull#2" in the right hand list, and click the apply button.

Review

If you select the review button, an HTML summary is opened that displays which shell parts have been included in the Euler coupling, the thickness and direction for these shell parts, and also the thickened shell joined matrix, that shows which shells are included in the normal calculation.

You can also view the thickened shells by switching on "Shell thickness" in the Plots panel, and choosing "Artificial Shell Coupling Thickness" in the Shell Thickness Settings panel.

14.3.5. Interactions - Euler/Lagrange (3D) - Weak Coupling

Within Autodyn-3D, there is a facility for the application of a pressure field in 3D space to Lagrange type parts as a boundary condition. The pressures in 3D space are contained in a sequence of Field files, the format of which is described below. Also included is an additional feature that enables you to create these fields when using either the Ideal Gas or the Multi-material Euler processors. Weak Lagrange Coupling can be seen as equivalent to the current Euler-Lagrange coupling but without the feedback from the Lagrange type parts back into the Euler part. This weak coupling assumes that any deflections in the Lagrange model are small, and should only be used in such cases.

The current implementation only considers the possibility of mapping pressure data, and applies this to the Lagrange boundary surfaces as nodal forces. The pressures will be applied to all wetted surfaces of Lagrange type subgrids within the volume described by the pressure files. The advantage over the current Euler-Lagrange coupling is that multiple Lagrange calculations can be carried out on one coupled Euler calculation, thus saving analysis time. Also, some speedup would be expected over a normal coupled solution for a single Euler and then Lagrange calculation.

Creating the Field files

The *Field* files are created as a field edit when an Ideal Gas or Multi-Material Euler type simulation is being carried out. You should click **Output** on the navigation bar and open the **Field** output part of the dialog panel. Under this menu there are three options, namely **Cycles** , **Times** and **Set_part** . The cycle and time options work in the traditional manner of all edits in Autodyn and enables you to select the frequency of the Field file output. The third menu, set_part, enables you to select which part the pressure data will be taken from, and the I, J and K range. The file format only allows one block of data to be written. If you do not supply this information, then the entire range of the first Euler part found will be written to the Field file. A geometry data field is only produced when a problem is started or restarted, and this cycle coincides with a field edit.

Using the Weak Interactions

The coupling option **Weak Coupling** is included under the Euler/Lagrange Interaction tab after **Interaction** is selected from the navigation bar. The Euler-Lagrange/Shell interactions need to be

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

set to chose which Lagrange type part are included in the coupling, and how shells interact. Under the weak coupling settings, the **Ambient Pressure** should be entered. This pressure is removed from the Field file pressure, i.e.

$P_{APPLIED} = P_{FIELD} - P_{AMBIENT}$

Also the Field file set need to be selected by using the file browser to select the first in the required set.

Notes on the Mapping

Autodyn-3D interpolates linearly in time the pressure data from two Field files, the current time of the simulation being in-between the times of the fields. When a field file is read into the solver a mapping is created. This is a lookup table for each cell face in the Lagrange model that is activated pointing to a cell in the pressure field. If a geometry data field is not present in the Field file then the existing mapping from the previous field is used. The mapping is created using one of two possible geometry choices for the Lagrange type parts:

1. The initial coordinates of the Lagrangian parts (i.e. at time equal to zero). With this option, a geometrical remap is performed only at cycle zero, and during the analysis each Lagrangian face will use the pressure of the same Euler element from the different Field files.

2. The current coordinates of the Lagrangian parts. With this option, a continuous geometrical remapping takes place during the analysis and therefore is more expensive.

The interpolation in space is the same as for the current Euler-Lagrange interactions; the nearest cells center to the face center is used.

Field file format (.ADF FILES)

The ADF (Autodyn FIELD) file format contains field information on a regular structured rectangular grid. Scalar data is cell centered. The naming convention of the files is:

IDENT_####.ADF FORMAT:'(A,A1,I4.4,A4)'

Where #### is an increasing integer number, starting at 0001.

Record 1	FILEID (CH*256)	Character string which can be used to identify the file
Record 2	NCYCLE, TIME, NTU, NMU, NLU	NTU is the time unit (1:µs 2:ms, 3:s) NMU is the mass unit (1:mg 2:g, 3:kg) NLU is the length unit (1:mm, 2:cm, 3:m)
Record 3	NI, NJ, NK	Maximum number of nodes in each of the three directions
Record 3+(NS-1)*2	ITEM_NAME (CH*256)	Name of the item to follow; that is, GEOMETRY or PRESSURE
Record 4+(NS-1)*2	The data attached to the item	Item specific - see below.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Record (LAST)	"END OF SCALAR FILE"

The data attached to the items are:

GEOMETRY

XC (1:NI)	X positions, in line with the I axis
YC (1:NJ)	Y positions, in line with the J axis
ZC (1:NK)	Z positions, in line with the K axis
	40000 + file counter

PRESSURE

PN(1:NI-1, 1:NJ-1, 1:NK-1)	Pressure of each cell. I varies fastest

A geometry data record is only needed if the geometry has changed. The presence of this field enforces a new mapping. Therefore, if the geometry is only present in the first field, then the mapping is only done at the beginning of the calculation. On the other hand, if the geometry is present in every file, then a remapping is done every time a new flow file is read. If a file does contain a geometry data record then Autodyn expects that it will occur before the pressure record. A pressure record must exist in all files.

Errors, Warning and other Outputs

Errors encountered during the mapping cause a generic error, stop the current simulation. A summary of the error is written to the message window. The following table summarizes the error codes:

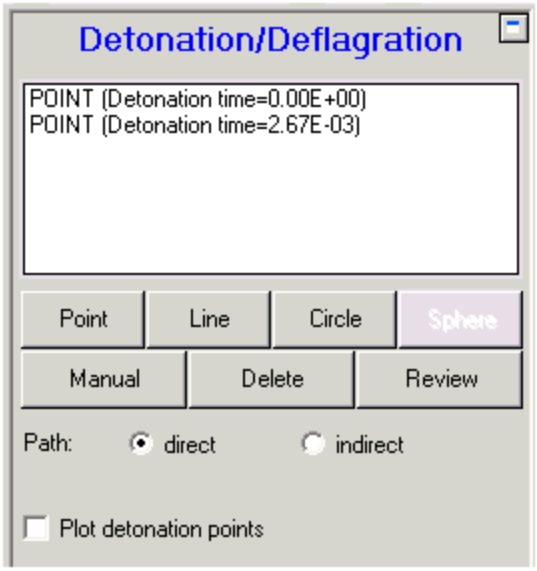
Too many scaler files	1
File does not exist	10000 + file counter
Expecting geometry data	20000 + file counter
No pressure data	30000 + file counter
Error in reading file	40000 + file counter

Errors and Warnings in the Field File writing routines will not stop the calculation, but will write an appropriate message to the message window.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information
of ANSYS, Inc. and its subsidiaries and affiliates.

Chapter 15: Autodyn Detonation (2D)



This window lets you set the locations where explosives will be detonated / deflagrated.

Detonations / Deflagrations

The box at the top of the window shows you a list of the current detonations / deflagrations you have defined. You can select from this list.

Point

Define a detonation point.

Line

Define a line of detonation.

Circle

Define a detonation circle (planar symmetry).

Sphere

Define a detonation sphere (axial symmetry).

Manual

Manually define initiation times for blocks of elements.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Delete

Delete the selected detonation definition

Review

Review information for all detonation definitions.

Path

Select whether you want detonation times to be calculated using direct or indirect paths.

Direct Path

Detonation times are computed from straight line paths from detonation points. This is the most accurate method, but is only valid if the straight line paths are through the explosive material.

Indirect Path

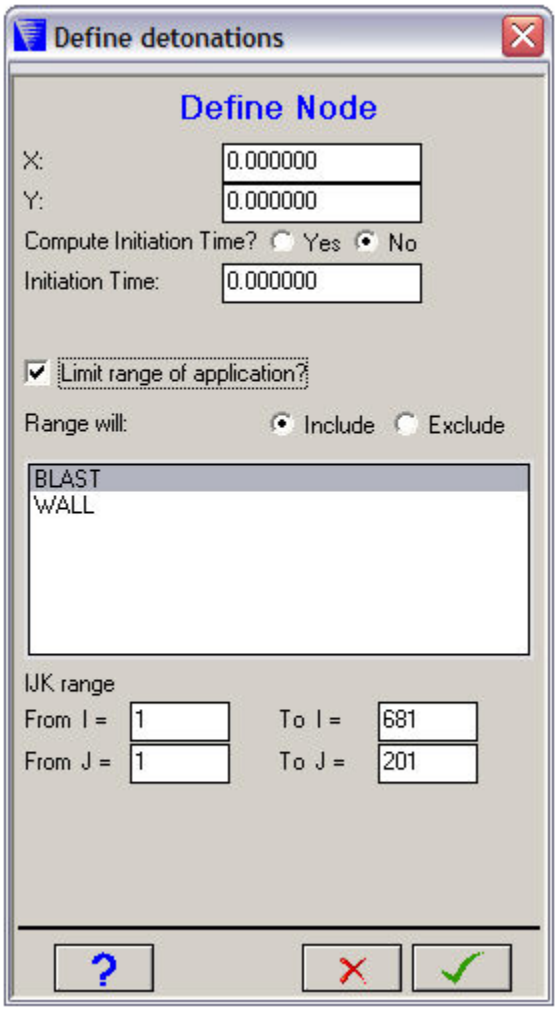
Detonation times are calculated from shortest paths through explosive material. Paths are automatically computed around inert materials (for example, wave shapers).

Plot detonation points

See the location of detonation points in the view panel.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

15.1. Detonation (2D) - Point



This window lets you define a detonation point.

X/Y Coordinates

The X/Y coordinates of the detonation point.

Compute Initiation Time?

Select whether you want to specify a detonation time, or have it computed by Autodyn.

If selected, Autodyn will compute the detonation time using paths through the explosive from other detonation points.

You must specify the initiation time for at least one detonation point, plane, cylinder or sphere.

Initiation Time

The initiation time (unless you chose to have it computed).

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Limit range of application?

Limits the range of application of the detonation point (an extra dialog is presented).

Range will Include / Exclude region

For each Part, you can specify an index range that is used to limit the range of application of the detonation point.

Select whether you want the index range to specify the included region or the excluded region.

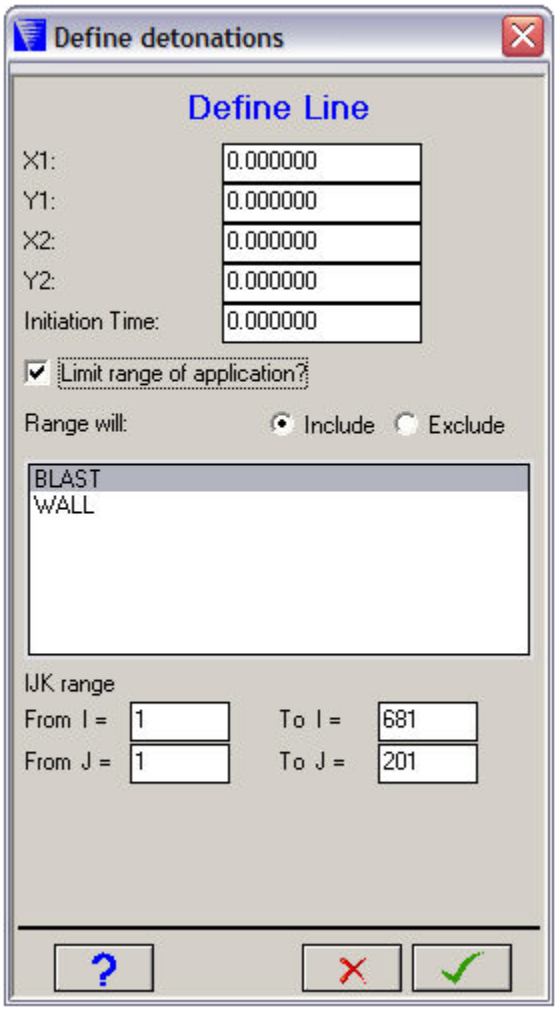
Part list

Lists all the Parts in your model. Select a Part in this window to limit its range of application.

IJ range

The IJ range you want to use to limit the range of application of the detonation point.

15.2. Detonation (2D) - Line



Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

This window lets you define a line of detonation.

X/Y Coordinates

The X and Y coordinates for the two end points of the line.

Initiation Time

The initiation time (unless you chose to have it computed).

Limit range of application?

Limit the range of application of the detonation point (an extra dialog is presented).

Range will Include / Exclude region

For each Part, you can specify an index range that is used to limit the range of application of the detonation point.

Select whether you want the index range to specify the included region or the excluded region.

Part list

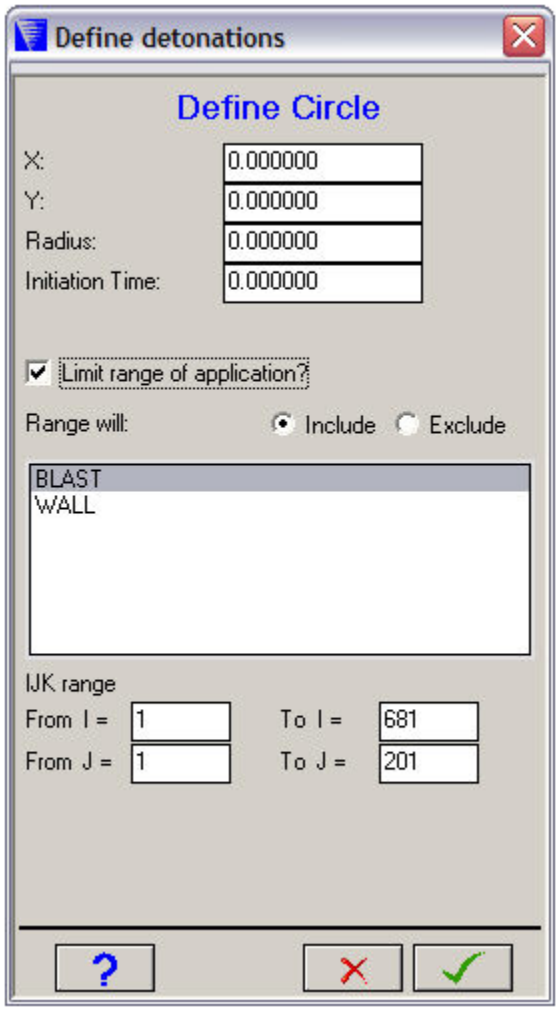
Lists all the Parts in your model. Select a Part in this window to limit its range of application.

IJ range

The IJ range you want to use to limit the range of application of the detonation point.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

15.3. Detonation (2D) - Circle



This window lets you define a detonation circle (detonation propagates radially inward and outward from the surface of the circle).

Center (X,Y)

The X and Y coordinates of the center of your circle.

Radius

The radius of your circle.

Initiation Time

Enter the initiation time in this field (unless you chose to have it computed).

Limit range of application?

Limits the range of application of the detonation point (an extra dialog is presented).

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Range will Include / Exclude region

For each Part, you can specify an index range that is used to limit the range of application of the detonation point.

Select whether you want the index range to specify the included region or the excluded region.

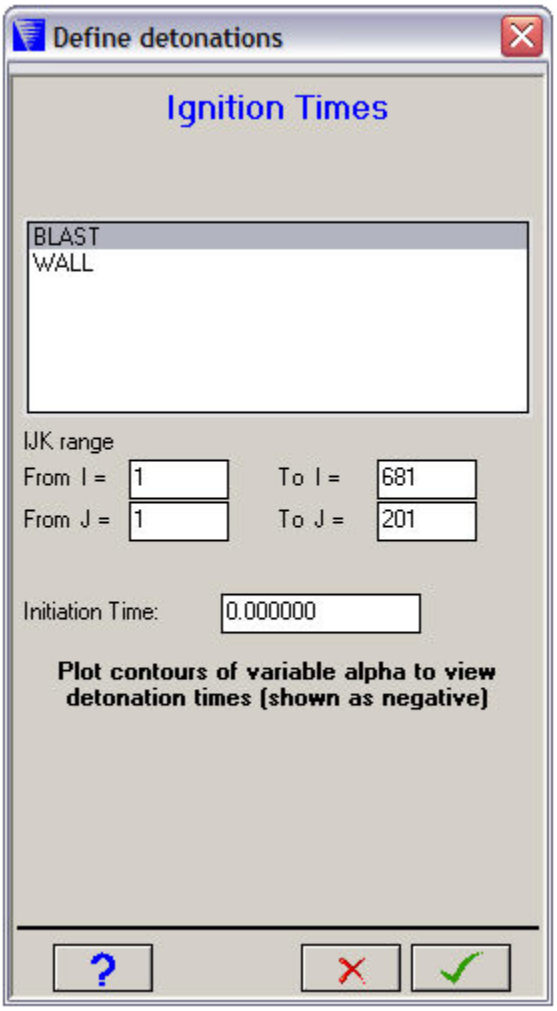
Part list

Lists all the Parts in your model. Select a Part in this window to limit its range of application.

IJ range

The IJ range you want to use to limit the range of application of the detonation point.

15.4. Detonation (2D) - Manual



This window lets you manually define initiation times for blocks of elements.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Part List

Lists all the Parts in your model. Select the Part you want to define initiation time for in this list.

IJ range

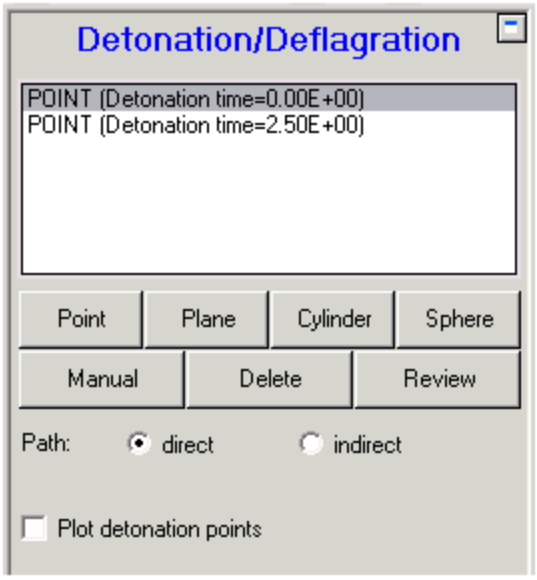
The IJ range for the block of elements to which you want to apply an initiation time.

Initiation time

The initiation time that will be applied to all elements in the block.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Chapter 16: Autodyn Detonation (3D)



This window lets you set the locations where explosives will be detonated / deflagrated.

Detonations / Deflagrations

The box at the top of the window shows you a list of the current detonations / deflagrations you have defined. You can select from this list.

Point

Define a detonation point.

Plane

Define a detonation plane.

Cylinder

Define a detonation cylinder.

Sphere

Define a detonation sphere.

Manual

Manually define initiation times for blocks of elements.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Delete

Delete the selected detonation definition.

Review

Review information for all detonation definitions.

Path

Select whether you want detonation times to be calculated using direct or indirect paths.

Direct Path

Detonation times are computed from straight line paths from detonation points. This is the most accurate method, but is only valid if the straight line paths are through the explosive material.

Indirect Path

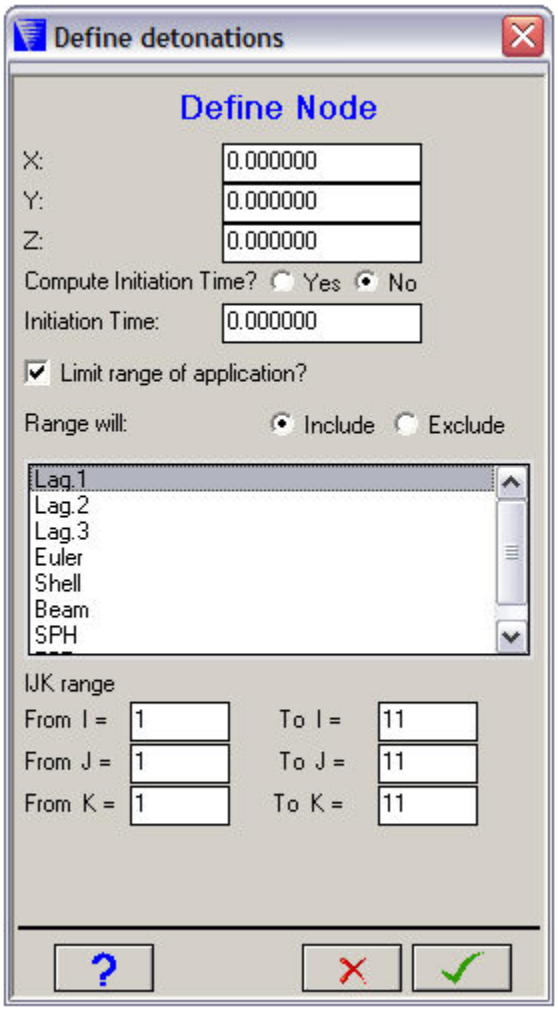
Detonation times are calculated from shortest paths through explosive material. Paths are automatically computed around inert materials (for example, wave shapers). Indirect Path cannot be used in models with SPH regions filled with explosive materials.

Plot detonation points

See the location of detonation points in the view panel.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

16.1. Detonation (3D) - Point



This window lets you define a detonation point.

X/Y/Z Coordinates

The X/Y/Z coordinates of the detonation point in these fields.

Compute Initiation Time?

Select whether you want to specify a detonation time, or have it computed by Autodyn.

If selected, Autodyn will compute the detonation time using paths through the explosive from other detonation points.

You must specify the initiation time for at least one detonation point, plane, cylinder or sphere.

Initiation Time

The initiation time (unless you chose to have it computed).

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Limit range of application?

Limit the range of application of the detonation point (an extra dialog is presented).

Range will Include / Exclude region

Specifies an index range that is used to limit the range of application of the detonation point.

Select whether you want the index range to specify the included region or the excluded region.

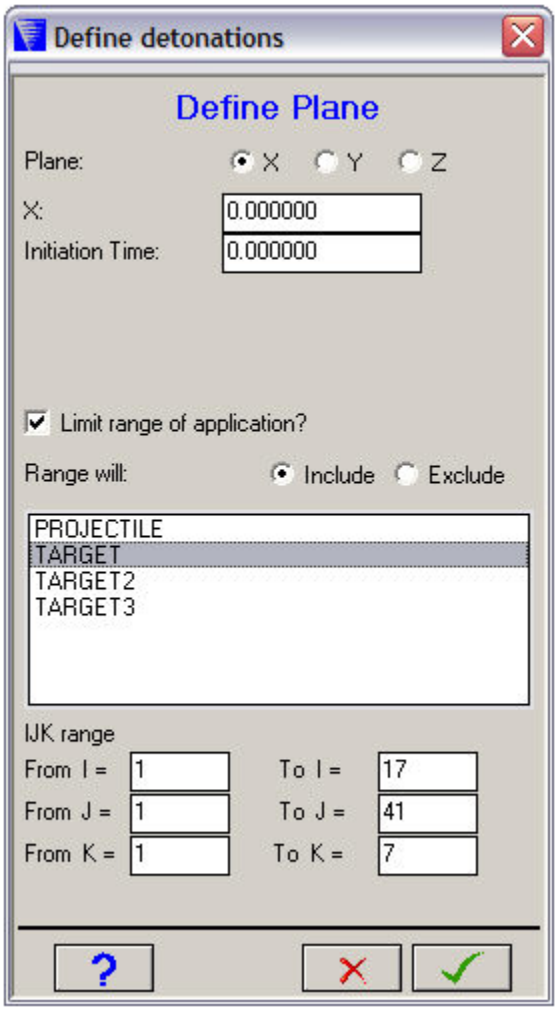
Part list

Lists all the Parts in your model. Select a Part in this window to limit its range of application.

IJK range

The IJK range you want to use to limit the range of application of the detonation point.

16.2. Detonation (3D) - Plane



This window lets you define a detonation plane.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Plane

Define an X, Y, or Z detonation plane.

X/Y/Z Coordinate

The X, Y, or Z coordinate of the plane.

Initiation Time

The initiation time in field (unless you chose to have it computed).

Limit range of application?

Limit the range of application of the detonation point (an extra dialog is presented).

Range will Include / Exclude region

For each Part, you can specify an index range that is used to limit the range of application of the detonation point.

Select whether you want the index range to specify the included region or the excluded region.

Part list

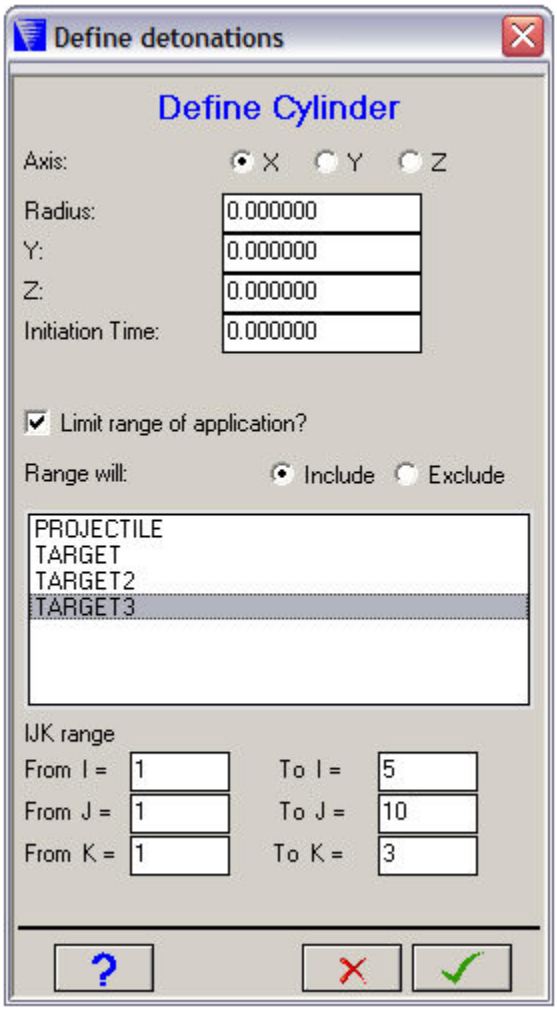
Lists all the Parts in your model. Select a Part in this window to limit its range of application.

IJK range

The IJK range you want to use to limit the range of application of the detonation point.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

16.3. Detonation (3D) - Cylinder



This window lets you define a detonation cylinder (detonation propagates radially inward and outward from the surface of the cylinder).

Axis

The axis (X, Y, or Z) to which the axis of symmetry of your cylinder will be parallel.

Radius

The radius of your cylinder.

Center Coordinates

The two coordinates for the center of your cylinder.

Initiation Time

The initiation time in field (unless you chose to have it computed).

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Limit range of application?

Limit the range of application of the detonation point (an extra dialog is presented).

Range will Include / Exclude region

For each Part, you can specify an index range that is used to limit the range of application of the detonation point.

Select whether you want the index range to specify the included region or the excluded region.

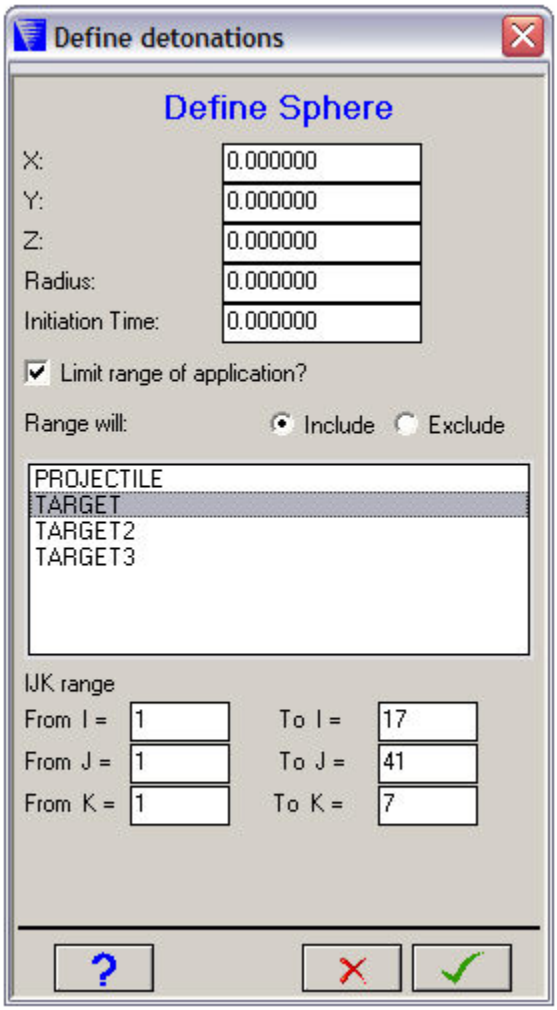
Part list

Lists all the Parts in your model. Select a Part in this window to limit its range of application.

IJK range

The IJK range you want to use to limit the range of application of the detonation point.

16.4. Detonation (3D) - Sphere



Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

This window lets you define a detonation sphere (detonation propagates radially inward and outward from the surface of the sphere).

Center Coordinates

The coordinates for the center of your sphere.

Radius

The radius of your sphere.

Initiation Time

The initiation time in field (unless you chose to have it computed).

Limit range of application?

Limit the range of application of the detonation point (an extra dialog is presented).

Range will Include / Exclude region

For each Part, you can specify an index range that is used to limit the range of application of the detonation point.

Select whether you want the index range to specify the included region or the excluded region.

Part list

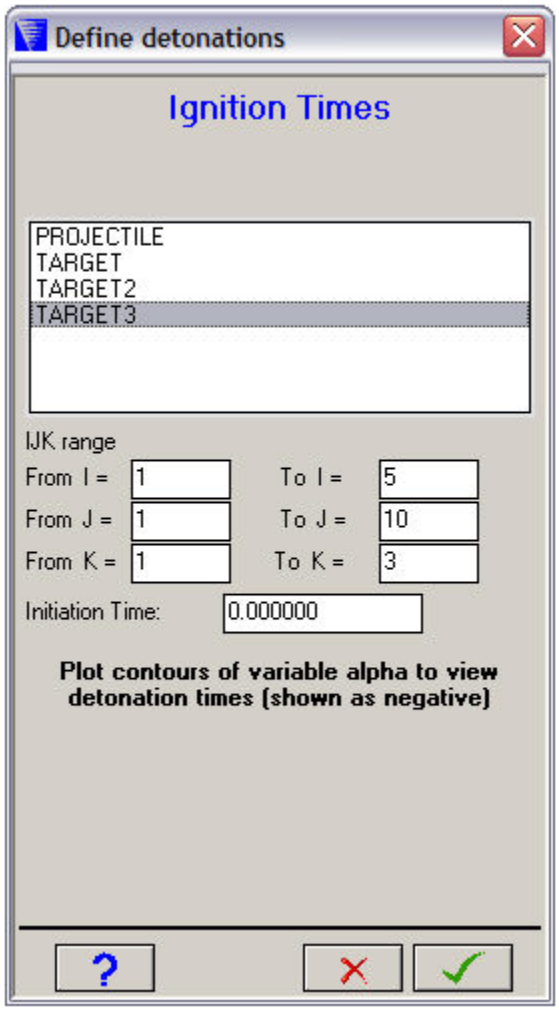
Lists all the Parts in your model. Select a Part in this window to limit its range of application.

IJK range

The IJK range you want to use to limit the range of application of the detonation point.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

16.5. Detonation (3D) - Manual



This window lets you manually define initiation times for blocks of elements.

Part list

Lists all the Parts in your model. Select the Part you want to define initiation time for in this list.

IJK range

The IJK range you want to use to limit the range of application of the detonation point.

Initiation Time

The initiation time that will be applied to all elements in the block.

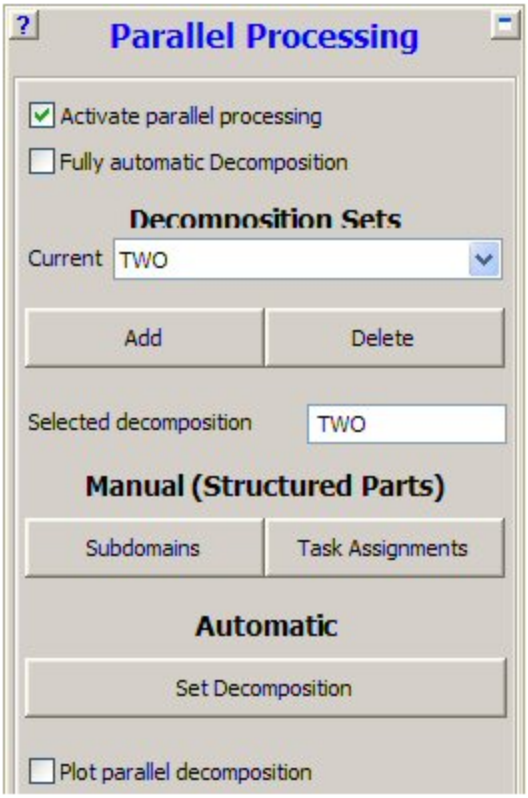
This feature allows you to manually define the detonation time for the cells. To see the detonation times, view the plot contours of alpha. The detonation time for each cell is shown as a negative number. If a cell shows a value of 0, it will start to detonate at cycle 0. Positive numbers between 0 and 1 correspond to the burn fraction for the cell.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

You can also view contour plots of pressure or internal energy to verify that energy has been added to the system.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Chapter 17: Autodyn Parallel Processing



This panel lets you set up your model for parallel processing. For additional information about setting up a parallel analysis, see the *Autodyn Parallel Processing Tutorial* on the Product Documentation section of the Ansys customer site.

Activate parallel processing

Check this box to activate parallel processing. If this box is checked, your model will be executed in parallel when you press

.

Fully automatic Decomposition

Check this box to activate fully automatic decomposition of models containing any combination of Euler and unstructured parts

Decomposition Sets

This input lets you define the domain decompositions you want to use for parallel processing by defining one or more Decomposition Sets.

Decomposition Sets are specific to your model.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

The box at the top shows the name of the current Host Decomposition Set. The pull-down menu for this box lets you change the current Decomposition Set to any other Set you have defined.

- **Add**

  Click this button to define a new Decomposition Set. Enter the name of your new Decomposition Set in the **Selected decomposition** field.

- **Delete**

  Click this button to delete the current Decomposition Set.

Manual (Structured Parts)

The structured Lagrangian solver is decomposed along its grid lines. These provide an easy and obvious method to decompose any structured Parts whilst maximizing the load balancing and minimizing the communication.

- **Subdomains**

  Click this button to decompose the Parts of your model into sub-domains (SPH parts are decomposed automatically).

- **Task Assignments**

  Click this button to assign the sub-domains to Tasks performed on each CPU.

Automatic

The automatic domain decomposition for unstructured parts is conducted using a recursive bisection method. This takes into account the geometric location of elements, the element type and the element density.

- **Set Decomposition**

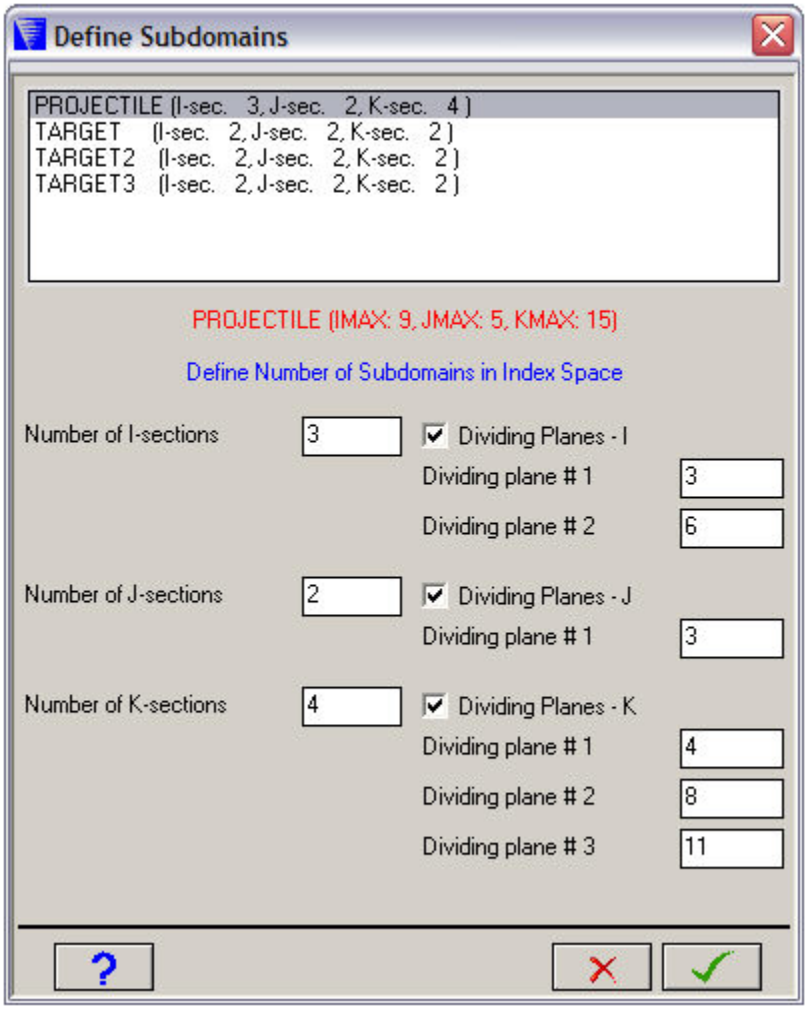
  Click this button to select the parts to be included in decomposition.

Plot parallel decomposition

Check this box to see the domain decomposition and task assignments for the current Decomposition Set.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

17.1. Parallel - Sub-domains



This window lets you decompose the Parts of your model into sub-domains for parallel processing (SPH parts are decomposed automatically).

You can decompose each grid-based Part in your model into smaller blocks, called sub-domains, that can be assigned to different tasks (on different hosts) for processing.

For each Part, these blocks are formed by defining planes in the I, J and K direction where cuts are made to form the sub-domains.

Parts List

The top box lists the model Parts that you can decompose for parallel processing. You can select a Part in this list.

Number of Sections

Enter the number of sections into which you want the selected part divided in I, J and K directions.

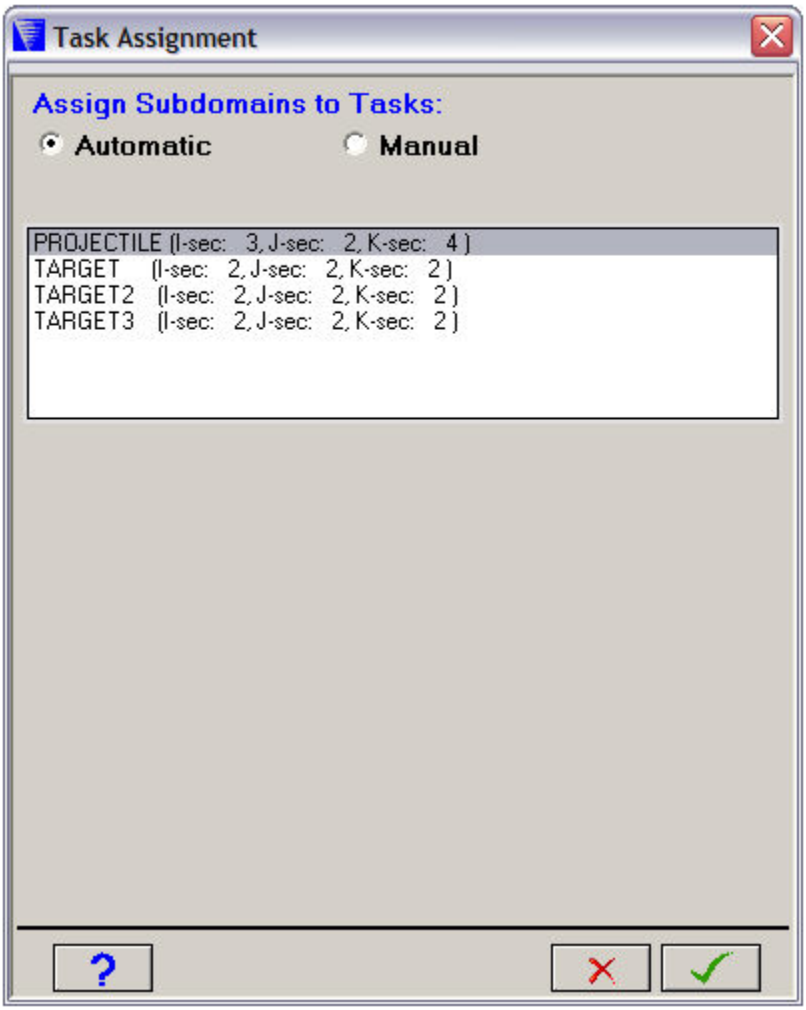
Dividing Planes

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Enter the I, J and K values for the dividing planes in the three index directions (the default values place the dividing planes evenly through the grid).

The example above would decompose the **PROJECTILE** Part into 24 sub-domains.

17.2. Parallel - Task Assignments



This window lets you assign sub-domains to Tasks performed on each CPU.

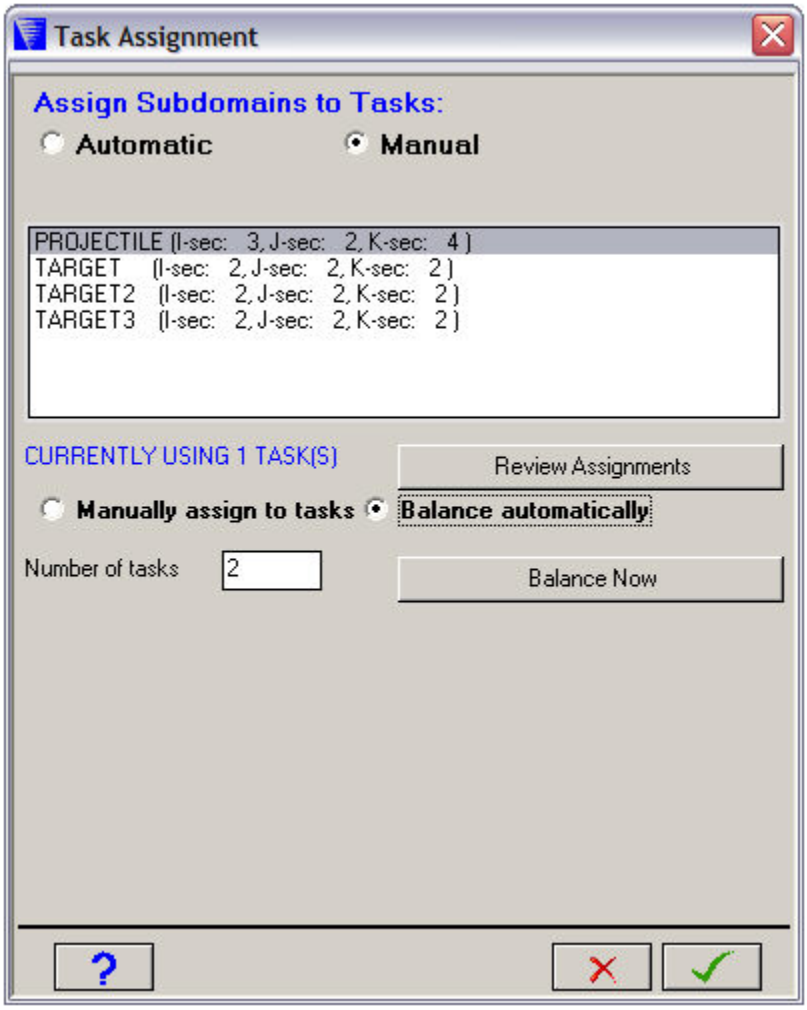
The efficiency of parallel processing is very dependent on distributing sub-domains among the tasks so that each task has approximately the same amount of work (computing) to do.

Automatic / Manual assignment

If you select **Automatic** assignment of Sub-domains to Tasks, Autodyn will automatically distribute sub-domains among the tasks, using load-balancing algorithms. You do not need to input any further data if you use this option.

If you select Manual assignment of Sub-domains to Tasks, further input dialog is presented:

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Parts List

The box at the top of the window lists the Parts whose Sub-domains you need to assign. You can select a Part in this list.

Review Assignments

Click this button to view a list of all Sub-domains and the tasks to which they are assigned.

Balance Automatically

- If you select this option, you can invoke the load-balancing algorithms use if you had selected **Automatic** assignment of the Sub-domains to Tasks, to distribute your Sub-domains over a specific number of Tasks.
- **Number of tasks**

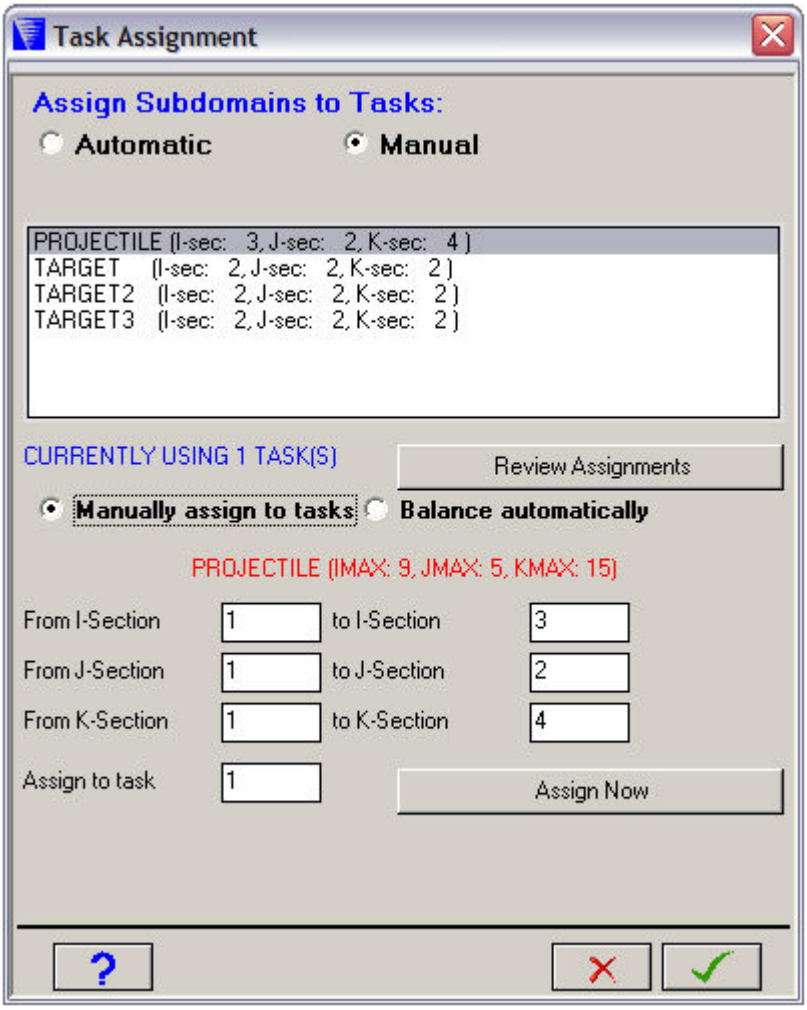
  Enter the number of Tasks over which you want to distribute your Sub-domains.
- **Balance Now**

  Click this button to perform the distribution.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Manually assign to tasks

- If you select this option, dialog is presented which lets you manually assign blocks of Sub-domains to a specific Task :



- **Block to be assigned**

  In the fields provided, specify the I, J, and K section range for the block of Sub-domains you want to assign to a Task.

- **Assign to Task**

  Enter the Task # to which you want to assign the block of Sub-domains in this field.

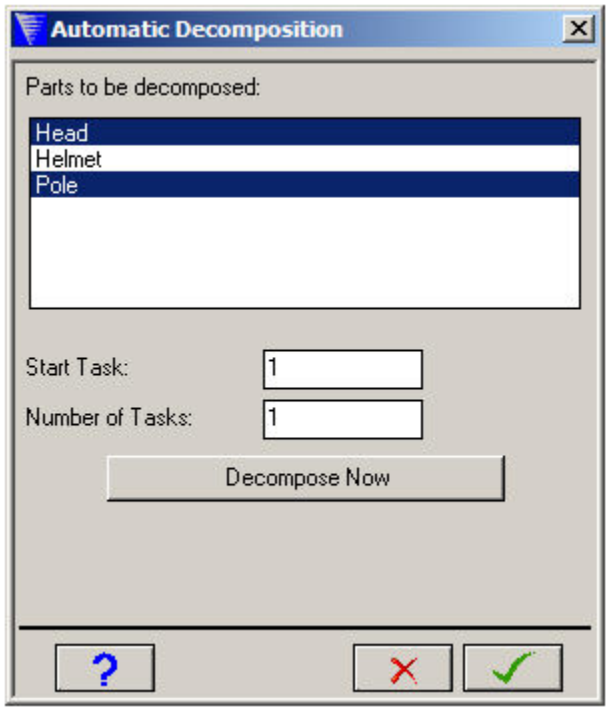
- **Assign Now**

  Click this button to assign the specified block to the specified Task.

  This procedure can be repeated as many times as you want, to assign blocks of Sub-domains to Tasks.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

17.3. Parallel - Automatic Decomposition



Parts to be decomposed

Select the parts to be decomposed from the list of unstructured parts and Euler parts (by default all unstructured parts and Euler parts will be selected).

Start Task

Enter the start task.

Number of tasks

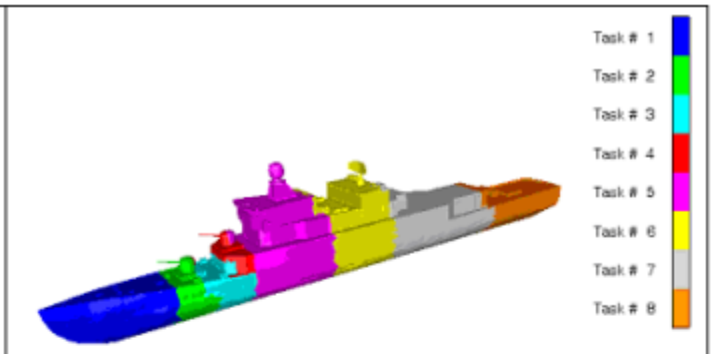
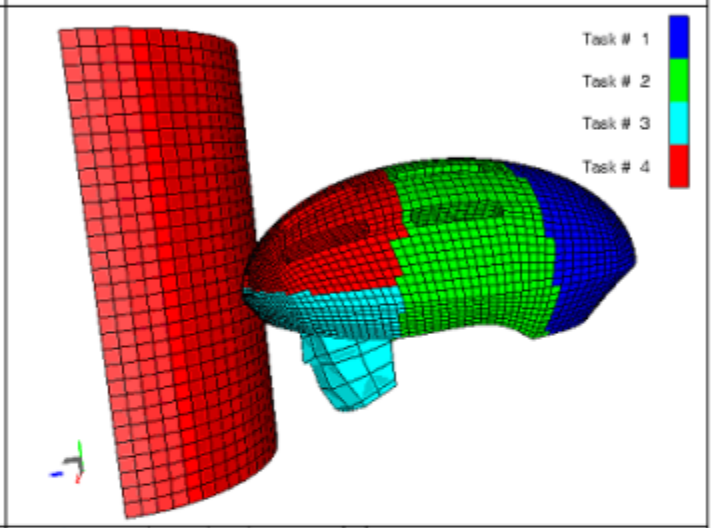
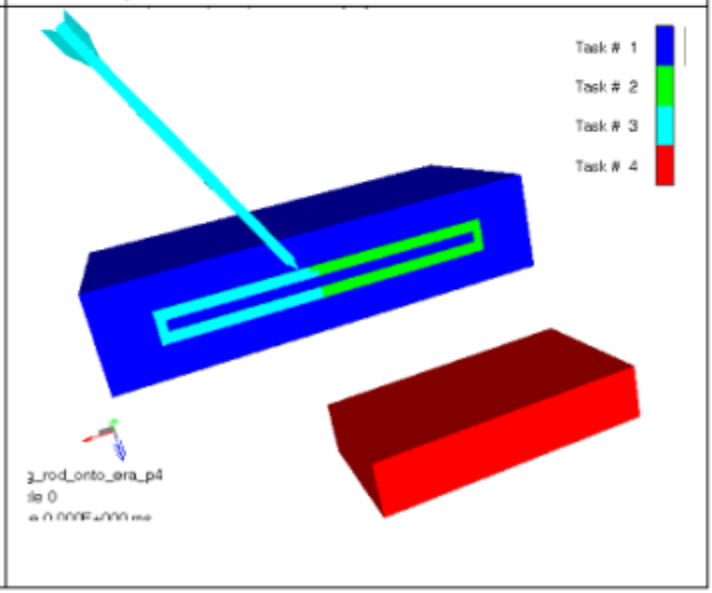
Enter the number of tasks

Decompose now

Click the *Decompose Now* button to automatically define the task on which each of the elements contained in the selected Parts will be assigned.

Examples

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Single unstructured Part • Select Part • Start Task = 1 • Number Tasks = 8	
All Parts Unstructured One rigid material • Select All Parts • Start Task = 1 • Number Tasks = 1	
3 Unstructured Parts 1 Euler Part (Structured) • Manual assignment of Euler Part to Task 1 • Automatic assignment of 2 unstructured Parts (rod and first target) o Start Task = 2 o # Task = 2 Automatic assignment of unstructured Part (second target) o Start Task = 4 o # Tasks = 1	

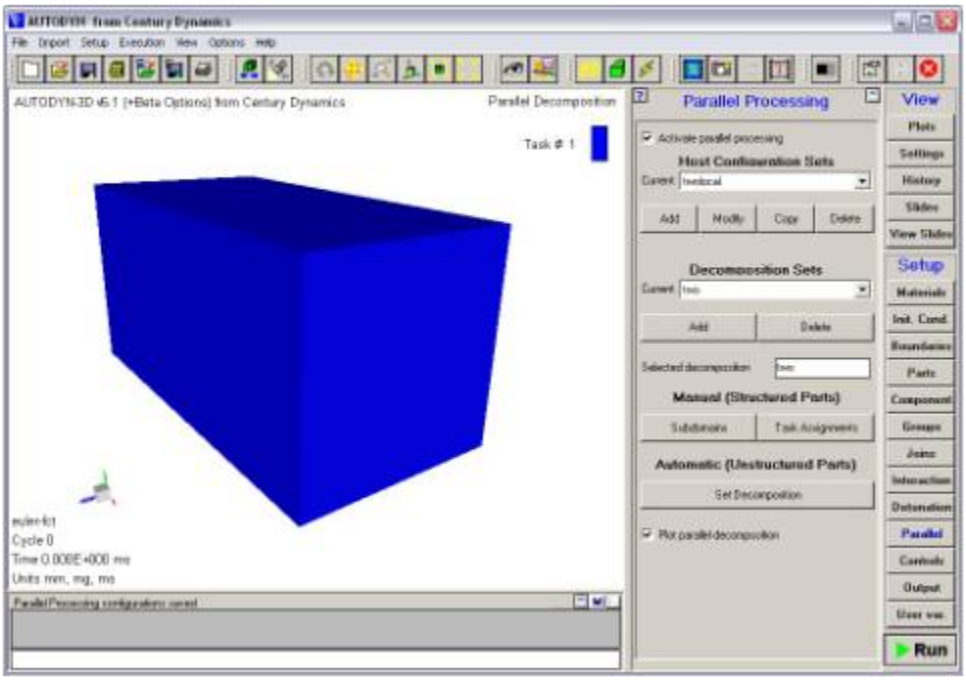
17.4. Parallel - Automatic Decomposition of Euler parts

The automatic decomposition algorithm also incorporates Euler parts, Ideal Gas as well as Multi-material, in addition to its ability to handle unstructured grids. This facilitates the decomposing of complicated Euler/Lagrange coupled models—you need only stipulate the number of tasks over which the model should be assigned and Autodyn will automatically produce a decomposition configuration with good load balancing qualities and minimal inter-processor communication. Furthermore, to further enhance

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

the efficiency of coupled calculations the subdomains of the Lagrange structure will be placed on the same processor, so decreasing the necessary inter-processor communication.

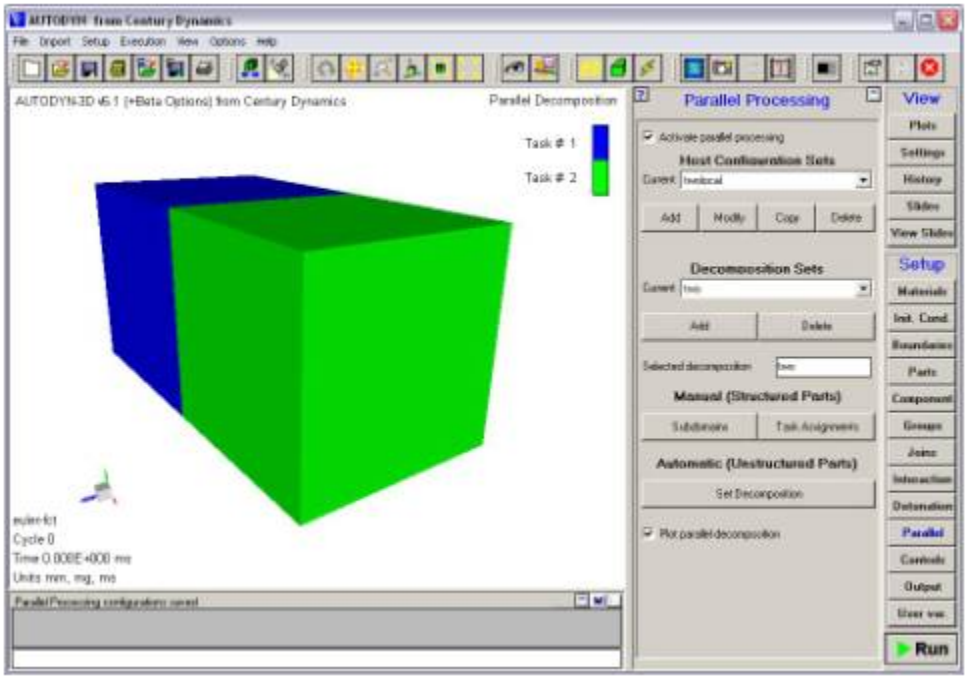
To automatically decompose an Euler model, go to the Parallel dialog panel as seen below, activate parallel processing and add a new decomposition set which in this instance is called 'two':



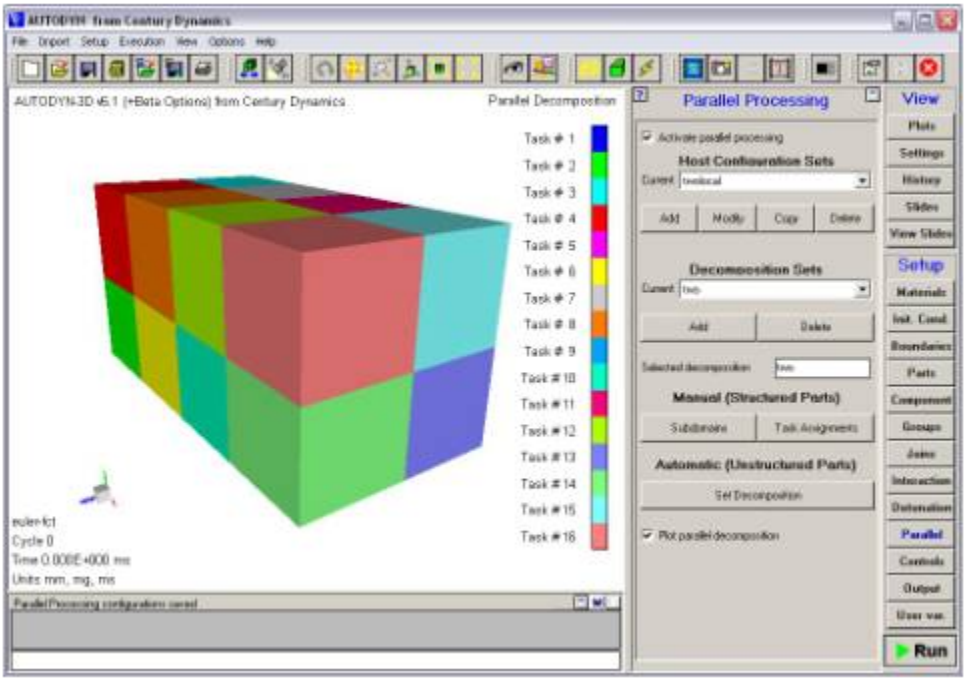
Click the **Set Decomposition** button and all Euler parts will be shown in the **Automatic Decomposition** dialog window:

Input the number of tasks you want to decompose the part over and click the **Decompose Now** button. In this case the number of tasks is two and the following decomposition can be viewed:

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

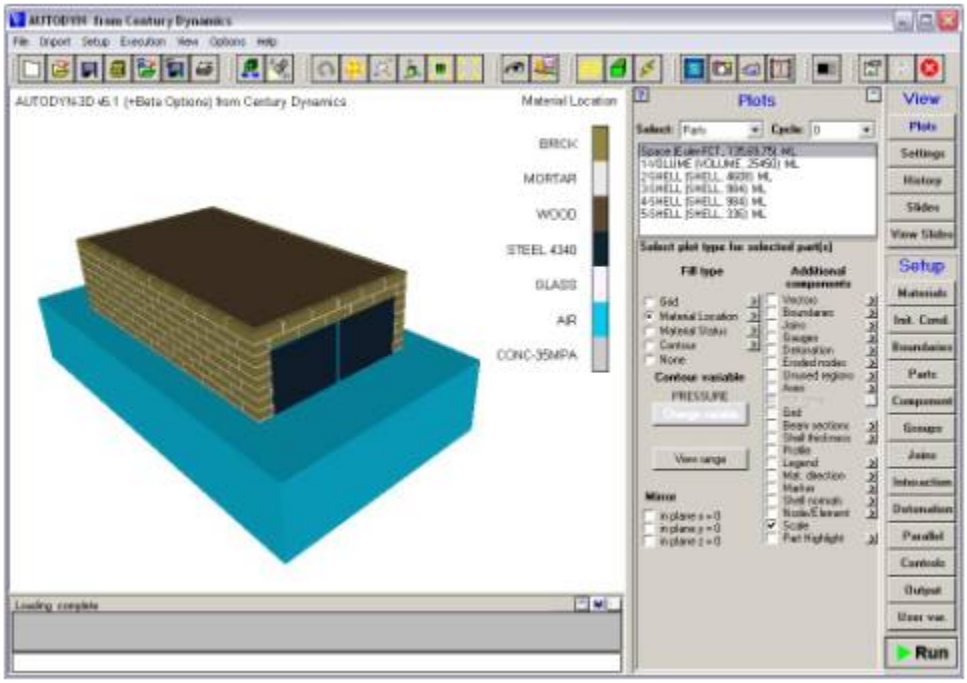


More complicated decompositions can also be easily achieved. Here the part is decomposed over 16 tasks:



As previously stated, the new algorithm allows easy decomposition of complicated coupled models which include Euler and unstructured Lagrange Parts. The model below contains a brick structure in which an explosion is occurring. The brick structure is modeled using Lagrangian parts, while the explosion in air is modeled using an Euler part. Here, only the lower region of the Euler part is shown for illustration purposes; the Euler part extends above the brick structure.

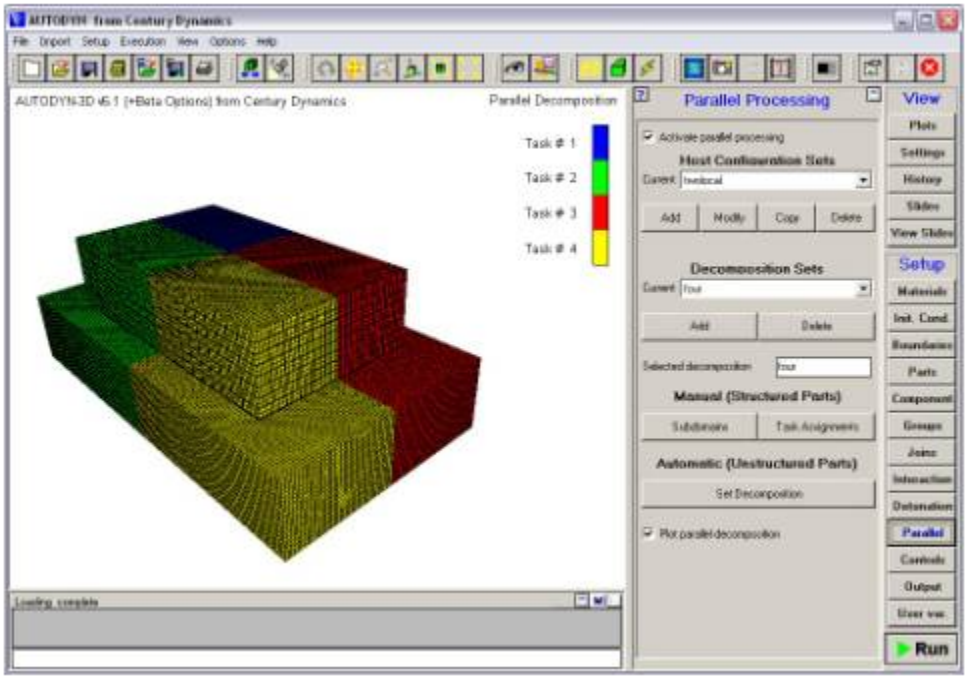
Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



The **Decomposition Set** window dialog will contain all of the parts and by default they will all be highlighted:

By choosing decomposing the model over 4 tasks, the various parts can be assigned to the available processors whilst ensuring efficient parallel computations:

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



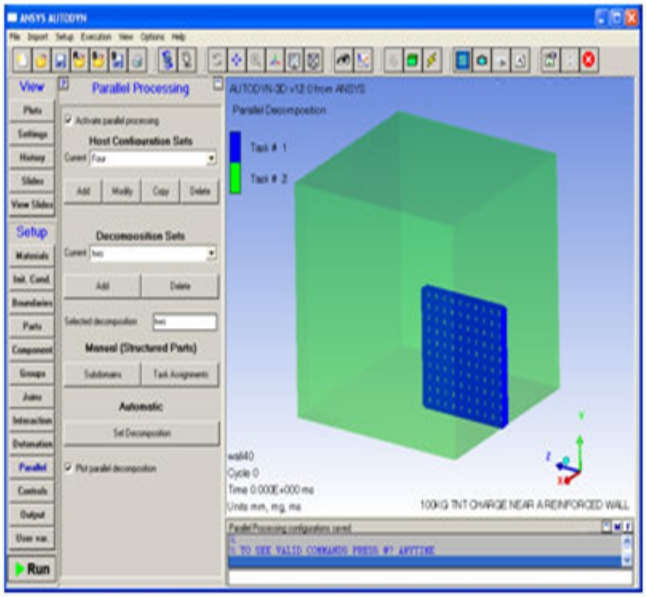
If you want to override this decomposition, then the Euler part can still be manually decomposed as before.

17.5. Parallel - Decomposition of beam parts

Autodyn-3D simulations including beam elements can be run in parallel.

Structured Beams

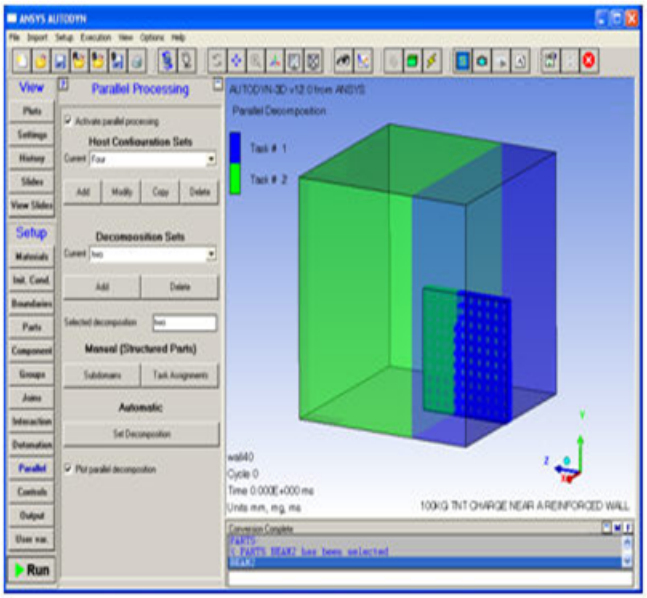
For the parallel computation, each structured Beam Part is considered as a single sub-domain and can be assigned to a specific task for the parallel simulation. Note that individual structured Beam Parts cannot be decomposed into multiple sub-domains.



Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

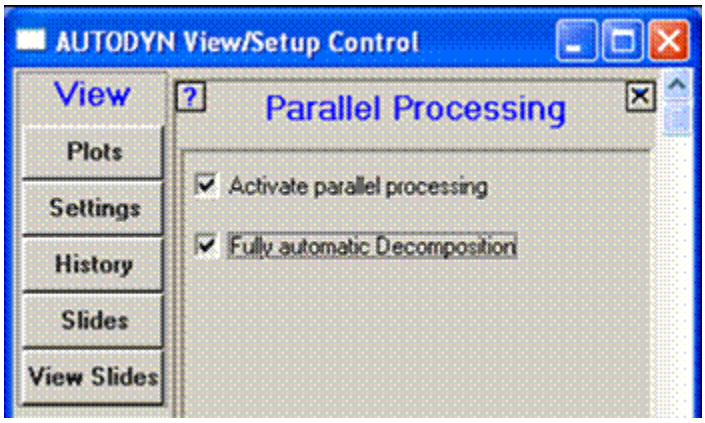
Unstructured Beams

For parallel computation unstructured Beam Parts can be automatically decomposed, similar to all other unstructured Lagrange Parts.



Note that parallel computations including beam elements can currently run with all join and contact options, with the exception of edge on edge contact. This later option is likely to be available in the near future.

17.6. Parallel – Fully Automatic Decomposition



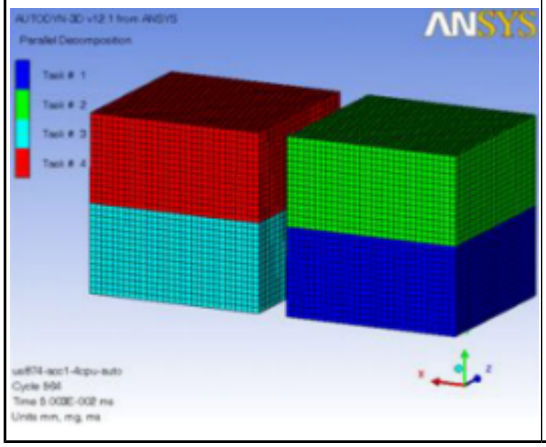
The Fully Automatic Decomposition option is only available for models containing any combination of Euler and unstructured parts. When the fully automatic decomposition option is chosen there is no further input required in terms of the parallel model set-up. On execution the model is automatically decomposed over the number of tasks specified in the Intel MPI appl file. At any point after the model has been initialized the resultant decomposition can be viewed using the plot option also found on the parallel panel.

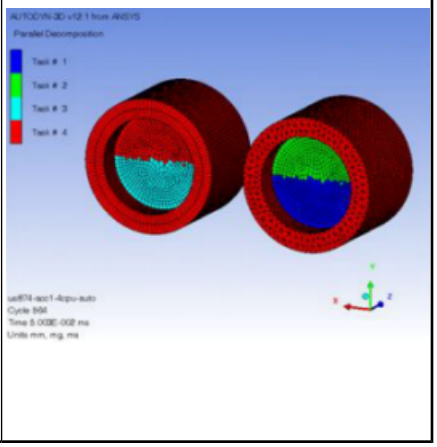
Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Example of Fully Automatic Decomposition of a coupled Euler/Lagrange model

Parallel model decomposed over 4 CPUs:

- 1 Euler Multi-material part
- 1 Euler Ideal Gas part
- 2 flexible unstructured parts
- 2 rigid unstructured parts (cylinders)

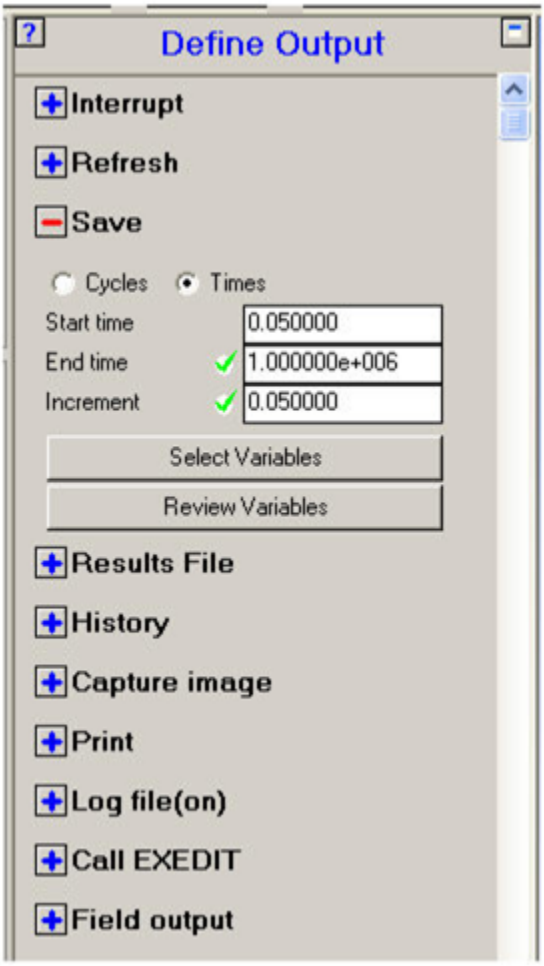




Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

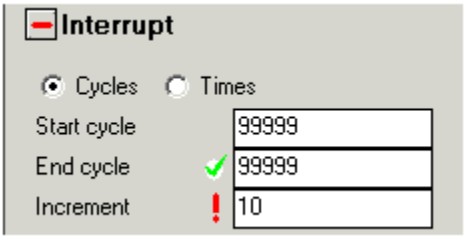
Chapter 18: Autodyn Output

Output



This window lets you set the type of output you want from your calculation.

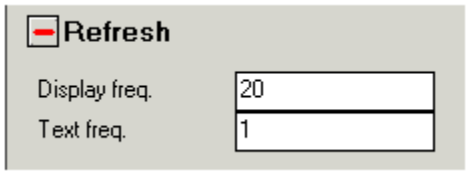
Interrupt



Set the frequency at which your calculation is interrupted (stopped) for you to plot, review and examine.

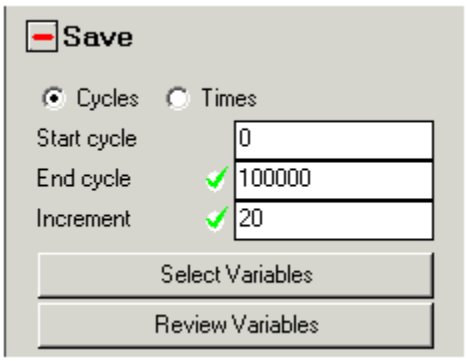
Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Refresh



Set the frequency at which your plot in the view panel is refreshed.

Save



The most important type of output you will want is data written to **Save** files.

Cycle / Times

Select whether you want data written at a set cycle frequency or time frequency.

Start cycle / time

Enter the cycle time that you want your first Save file to be written.

End cycle / time

Enter the cycle or time that you want your last Save file to be written.

If you are not sure how long you want to run your problem, enter a large number in this field, then you won't have to come back and change it later.

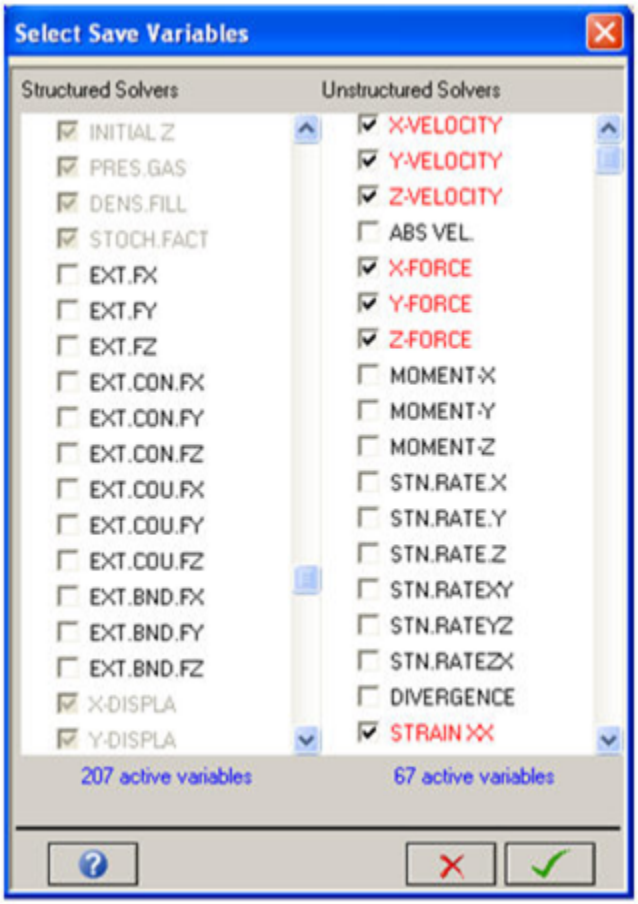
Increment

Enter the frequency you wish files to be written between the start cycle / time and the end cycle / time.

Select Variables

Click this button to select the structured or unstructured variables you want written to the **Save** file.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



For the structured solvers, the majority of variables is automatically selected and cannot be de-selected. This is because the memory for these variables is always allocated, even if the variable is not being used in a Part.

For the unstructured solvers, the full list of potential variables that can be used is presented. Only a small number of variables are pre-selected as commonly used variables. The selection of additional variables is up to you and must be made prior to execution of the model (at cycle zero). Only those variables selected will be available for output (contour, examine etc) Note that on execution, if a particular combination of element type and material requires a variable to be saved that has not been selected, Autodyn will automatically add this to the list.

Also note that the unstructured solves, some of the variables are never allocated physical memory, instead they are reconstructed from other variables that are allocated and stored in memory. An example of this would be the absolute velocity; this can be reconstructed from the nodal velocities hence does not require independent storage.

A similar list is also presented under the History, Variables option.

Note that the selection of additional postprocessing variables will result in higher memory usage and file sizes. If a significant number are selected, some degradation in runtime performance can also be expected.

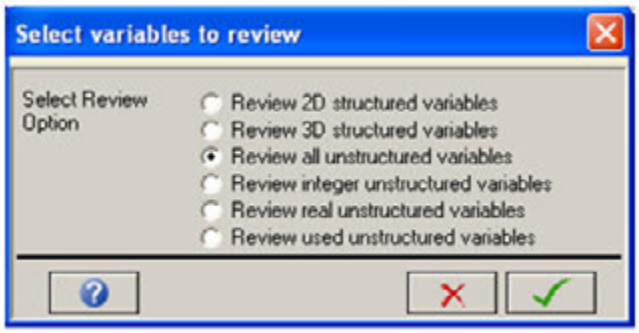
Click alongside any of the remaining output options to reveal the input fields for that option:

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

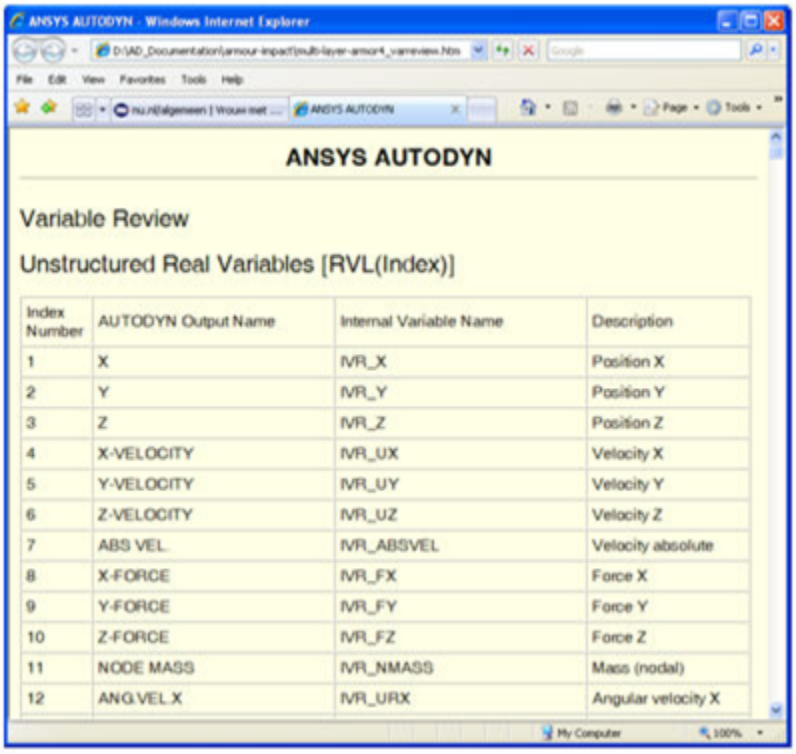
Review Variables

Select this button to create an HTML page of structured or unstructured variables.

After selection of the Review button first a selection window appears where the type of variable for review needs to be selected, e.g. 2D or 3D, structured or unstructured.



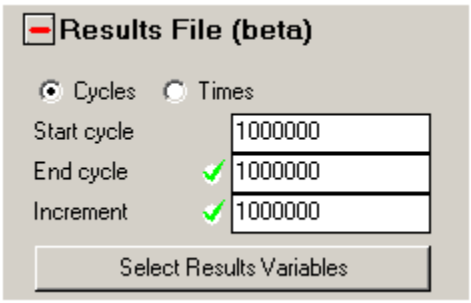
An HTML page will be created that shows a list of requested variables and their description.



Index Number	AUTODYN Output Name	Internal Variable Name	Description
1	X	IVR_X	Position X
2	Y	IVR_Y	Position Y
3	Z	IVR_Z	Position Z
4	X-VELOCITY	IVR_UX	Velocity X
5	Y-VELOCITY	IVR_UY	Velocity Y
6	Z-VELOCITY	IVR_UZ	Velocity Z
7	ABS VEL	IVR_ABSVEL	Velocity absolute
8	X-FORCE	IVR_FX	Force X
9	Y-FORCE	IVR_FY	Force Y
10	Z-FORCE	IVR_FZ	Force Z
11	NODE MASS	IVR_NMASS	Mass (nodal)
12	ANG.VEL.X	IVR_URX	Angular velocity X

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Results File

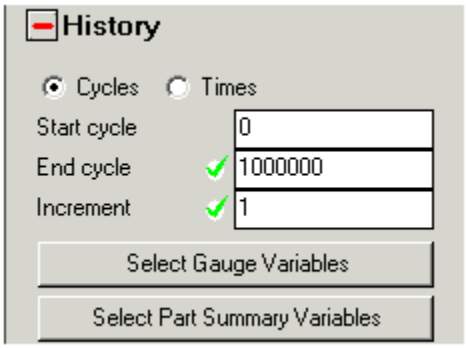


Set the frequency at which results files are written out.

Select Results Variable

Select the grid variables written to the result file.

History



Set the frequency at which history data is written.

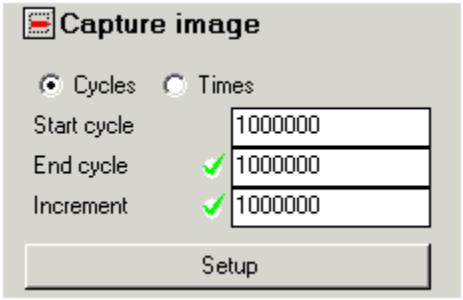
Select Gauge Variables

Sets which structured and unstructured variables to write to the history data file.

Select Part Summary Variables

Sets which part summary variables to write to the history data file.

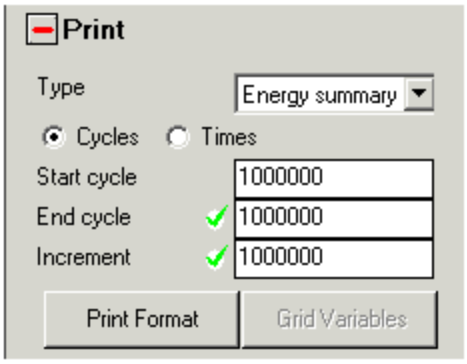
Capture image



Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Set the frequency at which automatic image captures are done.

Print



Set the frequency at which printed data is written.

Print Format

Sets the formatting of the information written to the print file.

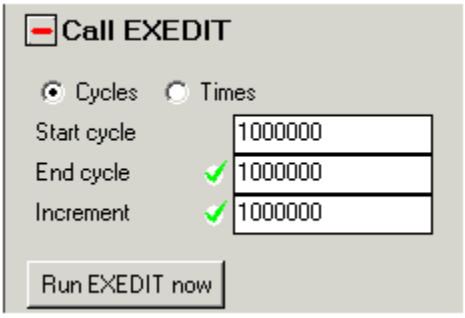
Grid Variables

Sets the grid variables written to the print file.

Log file

Turn the Log file on or off.

Call EXEDIT



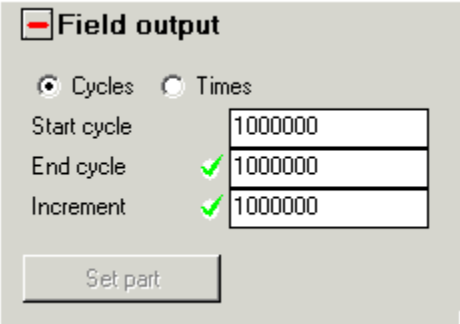
Set the frequency at which subroutine EXEDIT is called.

Run EXEDIT now

Click this button to call the EXEDIT user subroutine

Field output

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Set the frequency at which field output is written for weak coupling analyses.

18.1. Output - Part Summary Variables

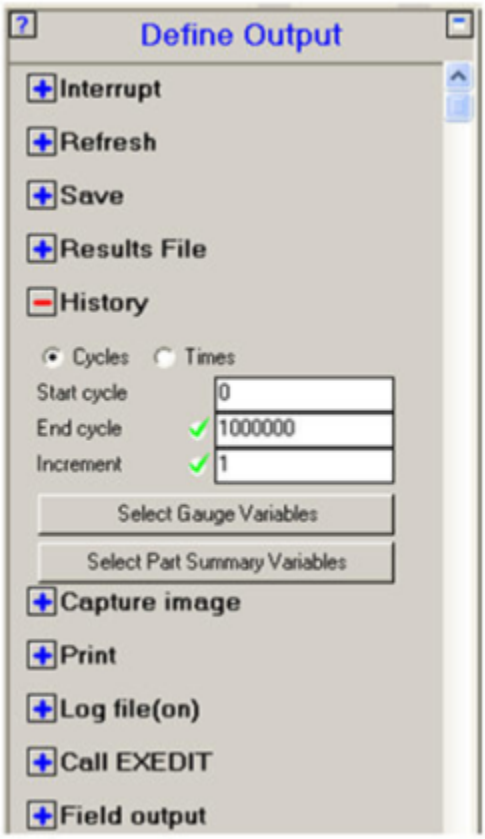
The Part summary variable includes three sets of optional data:

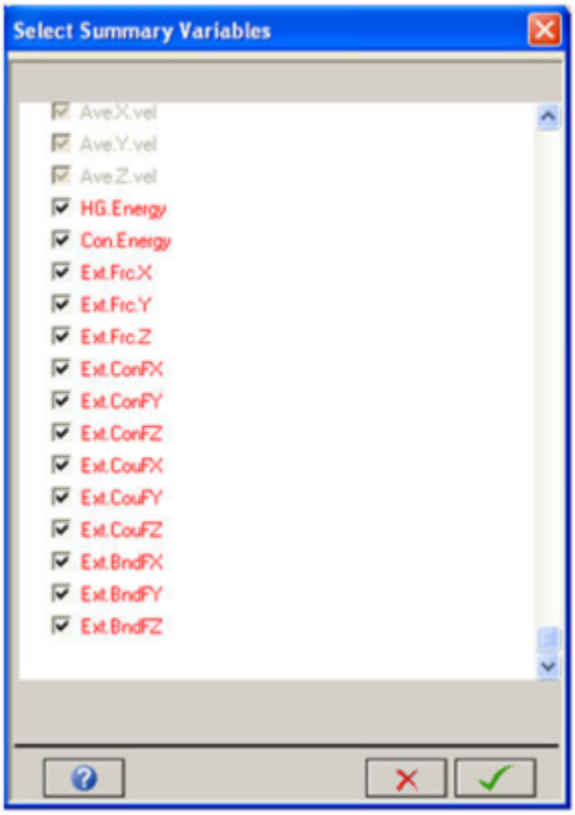
- Contact energy
- Hourglass energy
- External forces

This data is accumulated per Part and is available for review via the print file (using the energy summary option) or as time history data (using the Part summary option).

Note that by default, these additional variables are not available for history plotting. The variables must be activated using the Output, History, Select Part Summary Variables option.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.





Hourglass energy

The 2D and 3D Lagrangian volume and shell elements in Autodyn use hourglass control to remove zero energy modes of deformation. The forces of the hourglass control algorithms dissipate energy in resisting the zero energy modes of deformation.

In Autodyn, this energy is calculated and accumulated in the element internal energy. In version 6.1, the hourglass energy is now also accumulated independently to the internal energy to allow users to interrogate and assess the relevance of hourglass energy in a simulation.

Contact energy

Contact energy is accumulated on a per part basis as:

$$E^{n+1}_{npart} = E^{n}_{npart} + \Sigma_{Nodesonpart} F^{i}_{contact} \times \Delta x^{i}_{node} \tag{18.1}$$

For the purpose of energy accountancy, the contact energy is added to the work done on the system.

External forces

The resultant external forces on a Part can now be easily extracted from a simulation by selecting the appropriate components as Part summary variables. The components available are:

- **X/Y/Z contact forces**

  External forces applied to a Part from Lag/Lag contact

- **X/Y/Z coupling forces**

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

External forces applied to a Part from Euler/Lag coupling

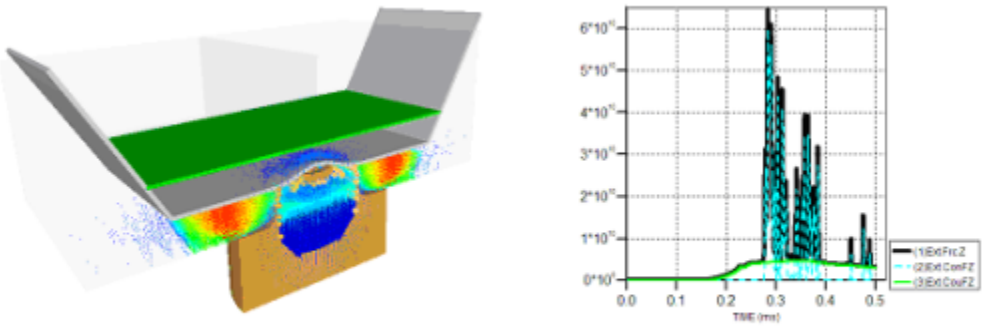
- **X/Y/Z boundary forces**

  External forces applied to Part from all boundary conditions

- **X/Y/Z total external forces**

  The sum of all external forces acting on a Part

Note: External forces are not extracted for polygon based Euler Lagrange coupling in 2D

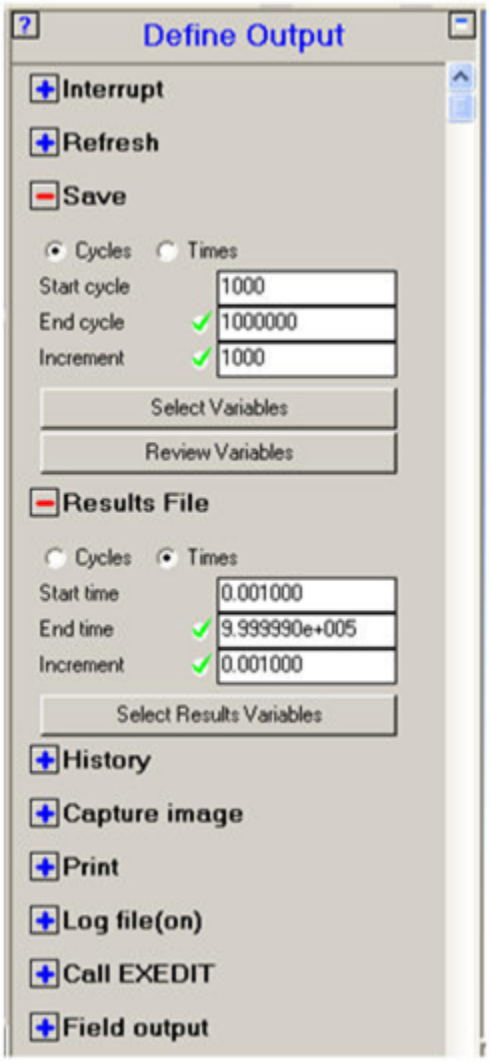


Example mine blast simulation highlighting source of loading (blast from coupling, impulse from soil ejecta) using new Part summary options

18.2. Output - Results Files

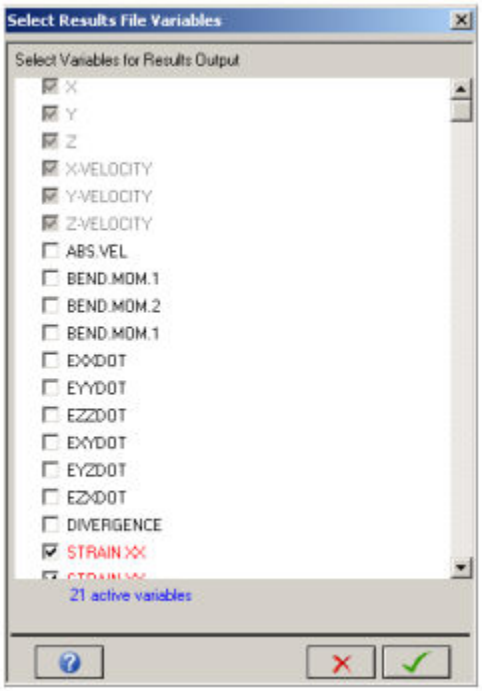
The capability to output results files, which contain a reduced amount of information, is available in Autodyn-3D. This allows the postprocessing of large simulations on computers with limited resources. Results Files cannot be used for restarting a problem. The frequency of the results files is independent of the SAVE files. The control for results file output is located in the Output panel, under Results Files:

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

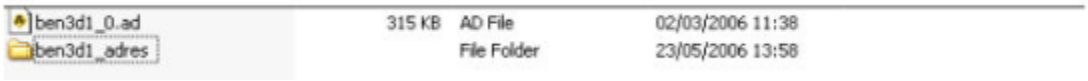


From here, the frequency of results file output and the variable to save into the file are selected. Note that only a limited set of results variables are selected by default.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



When a simulation is performed with the results file option switched on, a new directory will be created in the directory where the simulation file is located. This directory will have the name <*ident*>_*adres*. For example, the results files from a simulation with model name ben3d1_0.ad will be stored in a sub-directory with name `ben3d1_adres`.



Results files have extension adres, and the model name is appended with two numbers:

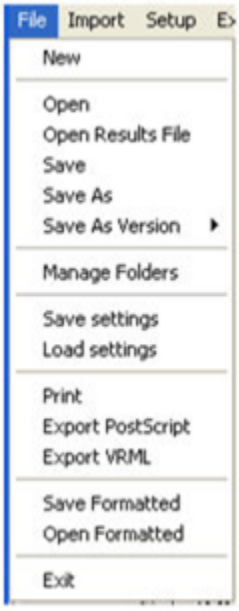
- The first number is the cycle from which the simulation began.
- The second number being the current cycle number.

For example, a results file `ben3d1_10_25.adres` indicates that the results file contains data at cycle 25 from the simulation ben3d1 that began at cycle 10.

Note that the start cycle for each sequence of results files is stored in a file <*ident*>_<*cycle*>.ad_base, for example ben3d1_10.ad_base. This base file is essential for post processing using results files and must not be deleted. The base file contains data required for producing graphic output that for efficiency is not repeated in each of the Results Files.

To use the results files for postprocessing of a simulation, the option under the File menu option should be:

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Once a results file has been selected, various options within Autodyn will be inactive, since no model alteration is possible. The cycle drop-down menu from the Plots panel is automatically updated to represent the cycles of results files that can be loaded. In order to switch back to full restart files simply load in a restart (cycle) file.

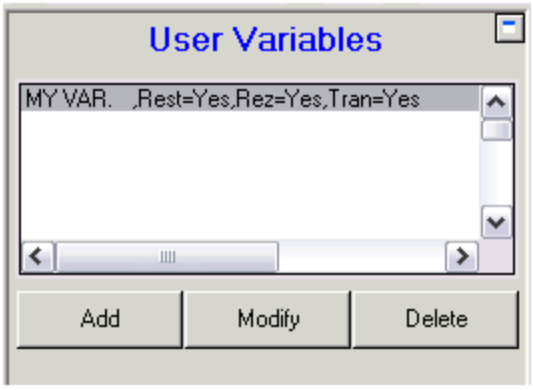
The address directory is designed to be independent of any other files. In order to share the results with colleagues simply transfer the directory.

Postprocessing using results files is significantly more memory efficient than from a restart file. Using results files, Autodyn only loads the data required to generate the requested Plots. The efficiency improvement will be problem dependant.

To use the results files to create an animation of the simulation, select the option "Scan for results files from this ident" from the Generate Multiple Slides (p. 153) panel.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Chapter 19: Autodyn User Variables



This panel lets you set up grid variables that you can use when programming your own user subroutines.

User Variable List

The top box lists the user variables you have currently defined. It shows the name you have assigned to the variable and whether or not the variable is written to restart (Save) files, rezoned and transported (in Euler). You can select a variable in this list.

Add

Adds a new user variable.

Modify

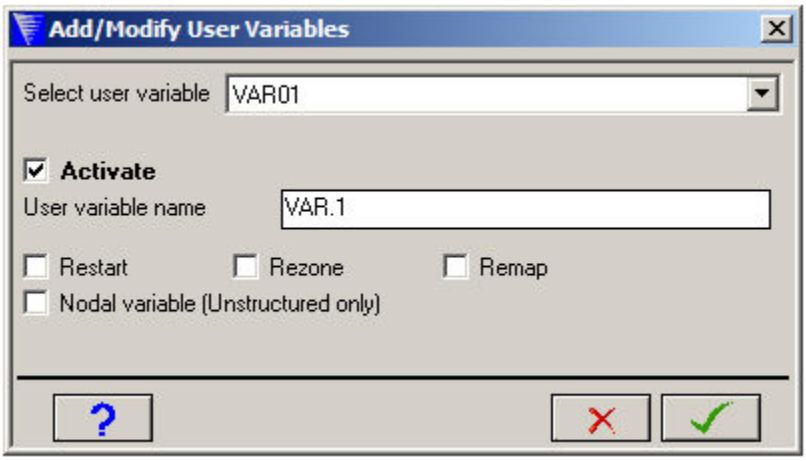
Enables editing of the selected user variable.

Delete

Deletes the selected user variable.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

19.1. User Variables - Add/Modify



This window lets you add a new user variable or modify the selected user variable.

Select user variable

You can define up to 20 user variables. Use this pull-down menu to select the user variable you want to define or modify.

Activate

Check this box to activate (define) the selected user variable.

User variable name

Enter the name you want to use for your user variable.

Restart

Check this box if you want the user variable to be written to the restart (Save) file.

Rezone

Check this box if you want the user variable to be included in the list of variables that are automatically rezoned.

Transport

Check this box if you want the user variable to be included in the list of variables that are automatically transported in Euler Parts.

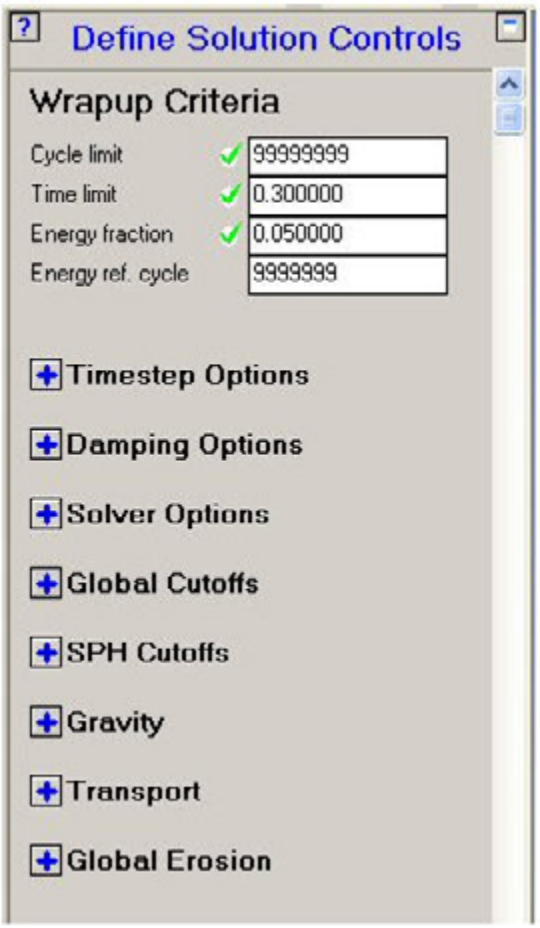
Nodal Variable

For the unstructured solvers, element and nodal quantities are stored independently. Check this box to define a user variable that is associated with a node, rather than an element for the unstructured solvers. Note that nodal user variable data can only be interrogated via examine and time history plots. Contour plots are not available.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Chapter 20: Autodyn Controls

Output



This panel lets you define solution controls for your model.

Wrapup Criteria

These controls are the only ones visible when you open this panel for the first time. This is because you must specify some of the parameters (all other control inputs have default values that are usually OK).

- **Cycle limit**

  Enter the maximum number of cycles you want your problem to run (enter a large integer if you don't want your problem to stop on a cycle limit).

- **Time limit**

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Enter the maximum time you want your problem to run (enter a large number - for example 1.0E20 - if you do not want your problem to stop on a time limit).

- **Energy fraction**

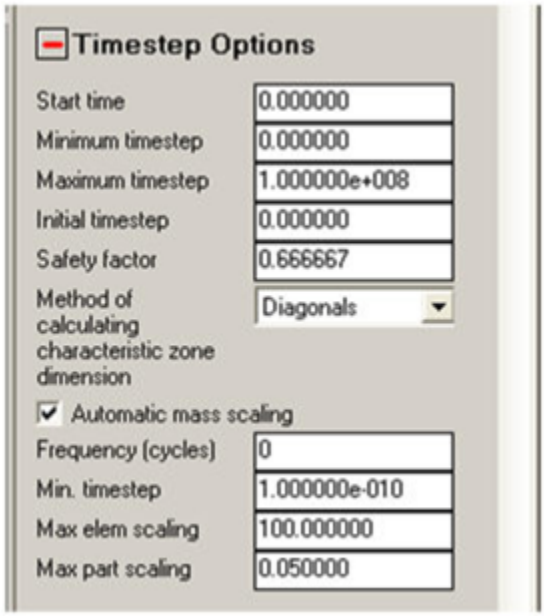
  Enter the energy fraction you want to use to stop your problem if the energy error gets too large. The default value of 0.05 causes your problem to stop if the energy error exceeds 5%.

- **Energy reference cycle**

  Enter the cycle # at which you want Autodyn to start checking for an energy error.

Press alongside any of the remaining options to reveal the input fields for that option :

Timestep Options



These options control the timestep Autodyn uses to run your problem.

- **Start time**

  Enter the start time for your problem.

- **Minimum timestep**

  Enter the minimum timestep you want to use.

  If the timestep drops below this value, your problem will be terminated.

  If you leave this field zero, the minimum timestep will be set to 1/10 of the Initial timestep.

- **Maximum timestep**

  Enter the maximum timestep you want to use.

  Autodyn will use the minimum of this value or the computed stability timestep.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- **Initial timestep**

  Enter the initial timestep you want to use.

  If you leave this field zero, the initial timestep will be set to 1/2 the computed stability timestep.

- **Safety factor**

  It is not wise to run problem at the stability limit, so a safety factor is applied to the computed stability timestep.

  Enter the stability factor you want to use.

  The default value (0.6666) is good for all problems, but you may want to increase this value to 0.9 in some cases to use a larger timestep (0.9 is a good value for most Lagrangian calculations).

- **Method of calculating characteristics zone**

  Three options are available for calculating the effective zone size used in the timestep calculation. By default, all models will use a zone size calculation based on element diagonals. If you want to obtain improved efficiency, especially in 3D simulations, the zone size calculation can be switched to one of two alternative options (based on either distances between opposing face centers or closest face centers).

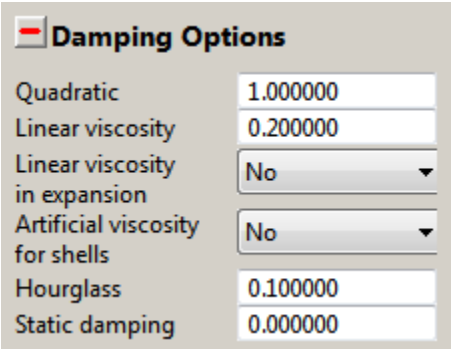
  To obtain the optimal time step for HEX solid elements the method of calculating the characteristic zone dimension needs to be set to Opposing Faces option instead of the default Diagonal option.

  Experience to date has shown that these options can significantly improve the efficiency of Lagrange simulations, especially in 3D. However, in certain circumstances when cells become highly distorted instabilities have been observed causing the calculation to terminate with high energy errors. The correct choice of erosion strain can reduce these problems. It is therefore recommended that users only utilize either of these non-default zone_size options if efficiency is critical.

- **Automatic Mass Scaling**

  Mass scaling is an artificial (numerical) mechanism for increasing the CFL (Courant-Friedrichs-Lewy) timestep of individual elements that govern the maximum allowed timestep of explicit transient dynamic solutions in Autodyn. Increasing the timestep has the obvious benefit of reducing the number of cycles required to run a simulation to a given point in time. Educated use of this option can therefore result in significant improvements in efficiency.

Damping Options



Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

These options control the magnitude of damping applied to your problem.

The default values are best for most problems.

Be very careful if you increase these values, they can significantly change your solution.

- **Quadratic viscosity**

  Enter the coefficient of quadratic viscosity you want to apply to shocks (compression waves).

- **Linear viscosity**

  Enter the coefficient of linear viscosity you want to apply to shocks (compression waves).

- **Linear viscosity in expansion**

  Choose whether you also want to apply the linear viscosity in expansion.

- **Artificial viscosity for shells**

  Choose whether to apply artificial viscosity to all structured shells in the model. The linear and quadratic coefficients used are those specified above. You can apply artificial viscosity to an unstructured shell using the **Solver Options** in the unstructured **Parts** menu.

- **Hourglass damping**

  Enter the coefficient of hourglass damping that you want to apply to Lagrangian elements.

- **Static damping**

  A static damping constant may be specified which changes the Autodyn calculation from a dynamic solution to a relaxation iteration converging to a state of stress equilibrium. For optimal convergence, the value chosen for the constant, R, may be defined by:

  ```
  R=2*timestep/T
  ```

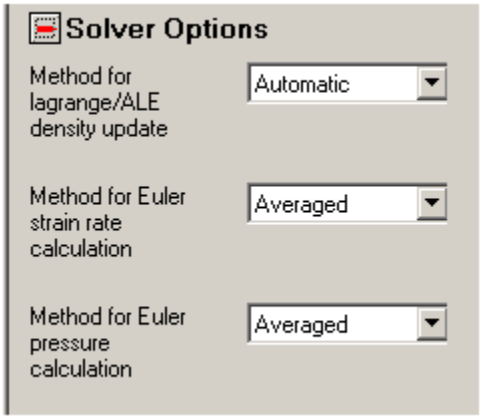
  where `timestep` is the expected average value of the timestep and `T` is longest period of vibration for the system being analyzed.

  Note:

  An Autodyn model that shares a setup with an Explicit Dynamics system that has more than one simulation step will have the static damping coefficient set per step, which is not visible in the Autodyn GUI. In this case, the static damping constant in the Autodyn GUI will display a value of 1e6 to indicate that the static damping coefficient is controlled by the simulation step data. In order to override the per step static damping constants, input the desired static damping constant into the static damping input field in the Autodyn GUI. The constant must be less than 1e6 for it to override the per step data.

Solver Options

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



- **Method for Lagrange/ALE density update**

  The element density can be updated each cycle using one of two methods; incrementally from the change in volume relative to the last cycle or recalculated based on the current mass and volume of the element. If the incremental change in volume over a cycle is small, it can be more accurate to use the incremental approach rather than the total approach to updating the density.

  Hence the options are:

  – **Automatic**

    Autodyn decides whether it needs to do an incremental update or not based on the change in volume.

  – **Incremental**

    Force Autodyn to always use the incremental update.

  – **Total**

    Force Autodyn to always recalculate the density.

- **Method for Euler Strain rate calculation**

  For multi-material Euler there are two methods to compute strain rates:

  – **Weighted.**

    For multi-material Euler analysis, involving materials with strength, strain rates need to be calculated to compute the increment in strain from cycle to cycle. The strain rates are derived from the velocity gradients, which are obtained by an element surface integration of the velocity. Using the Weighted option the face velocity that is used in the integration is a density weighted average of the element velocity on both sides of the face.

    This is the default option and provides the most accurate strain rate calculation.

  – **Averaged.**

    With this option the face velocity used in the integration is a simple average of the element velocity on both sides of the face and provides less accurate results than the Weighted option. Until Autodyn

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Version 6.1 the Averaged option was the only option available, and can be used to reproduce old results. Also models that are generated before Version 6.1 will still run using the Averaged method.

- **Method for Euler pressure calculation**

  For multi-material Euler cells containing two or more materials, there are two methods for calculating the resultant cell pressure:

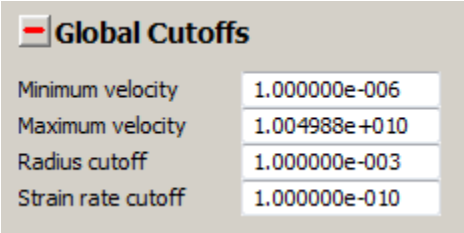
  – **Equilibrium**

  An iterative procedure is used for each material in a cell to establish a single pressure consistent with the equation of state and conditions of each material in the cell. This method is currently only applied if there are two materials in the cell and these materials are defined by ideal gas, or fully burnt JWL material. If this criterion is not met in a cell the pressure averaging option is used.

  – **Average**

  The resultant cell pressure is calculated from a weighted average of the pressure of the individual materials in the cell. The weighting algorithm takes account of both the volume fraction of each material and the relative stiffness of each material. This is the default method.

Global Cutoffs



These options let you set global cutoffs.

The default values are usually good if you are using the default units (mm, mg, ms). You may want to change one or more of these defaults if you are using some other set of units.

- **Minimum velocity**

  Enter the minimum velocity cutoff you want to use.

  If any velocity drops below this value it is set to zero.

- **Maximum velocity**

  Enter the maximum velocity you want to use.

  If any velocity rises above this value it is set to this value.

  This option is sometimes useful if your problem develops high velocities, which result in a small timestep, in regions that are not important to your solution.

- **Radius cutoff**

  Enter a value for the radius cut off for 2D axial models.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

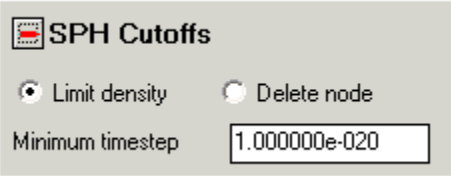
If a Lagrangian node is inside this radius at the start of your calculation, it is placed on the axis of symmetry.

If a Lagrangian node is outside this radius at the start of your calculation, it will not be allowed to come closer than this radius to the axis of symmetry as your calculation proceeds.

- **Strain rate cutoff**

  Enter a value for the minimum strain rate cutoff. If any strain rate drops below this value, it will be set to zero. The default is recommended for most analyses. For low speed or quasi-static analyses, it may be necessary to decrease this value.

SPH Cutoffs



These options let you set SPH cutoffs.

The default values are usually good if you are using the default units (mm, mg, ms). You may want to change one or more of these defaults if you are using some other set of units.

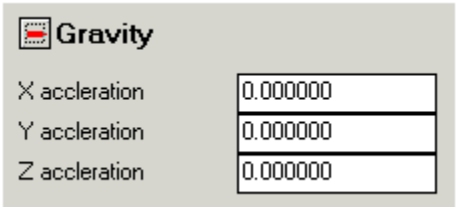
- **Limit density / Delete node**

  If the density of an SPH node drops below the minimum density or above the maximum density defined by the minimum and maximum density factor chosen for the material, you have the choice of either limiting the density to the cutoff density or deleting the node. Make your choice here.

- **Minimum timestep**

  Nodes that have a timestep below the minimum timestep will be deleted.

Gravity



This option lets you set gravitational components for your model.

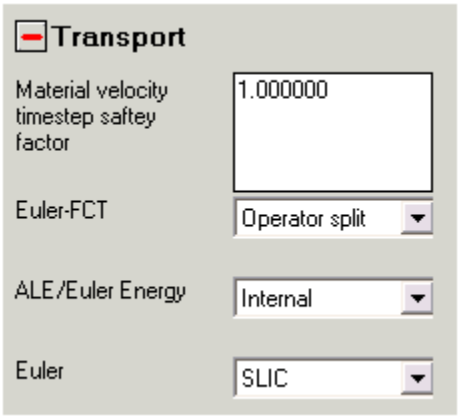
- **X / Y / Z acceleration**

  Enter the X Y and Z components of gravity in these fields.

  The gravitational acceleration components will be applied to all Parts present in the model.

Transport

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



These options relate to the transport of material in various Solvers.

- **Material velocity timestep safety factor**

  Enter the timestep safety factor for material velocities.

  In the velocity used to calculate the stability timestep is **c+u**, where **c** is the soundspeed of the material and **u** is its physical velocity.

  If **u** is of the order of or greater than **c**, then using **c + u/f** can give more stable results, where **f** is the safety factor.

- **Euler-FCT**

  Select the type of transport algorithm that is used for Euler-FCT calculations from this pull-down menu. The default setting is multi-dimensional where all three spatial dimensions are solved simultaneously which is more efficient and helps to maintain symmetric flow fields. The alternative is Operator Split where each spatial dimension is solved in turn.

- **ALE/Euler Energy**

  During a computational Euler cycle mass, momentum and energy is being transported (fluxed) from element to element. This option determines if total energy or internal energy is being transported.

  – **Total**

    This option conserves the total energy (kinetic + internal energy) during transport. This is the most robust and accurate option and is the default.

  – **Internal**

    This option conserves the internal energy during transport. In cases where the total energy transport doesn't provide an accurate answer, e.g. material overheating or when the kinetic energy is predominant, the Internal option might be chosen as an alternative.

- **Euler**

  This option determines which transport algorithm will be used for multi-material transport.

  – **SLIC**

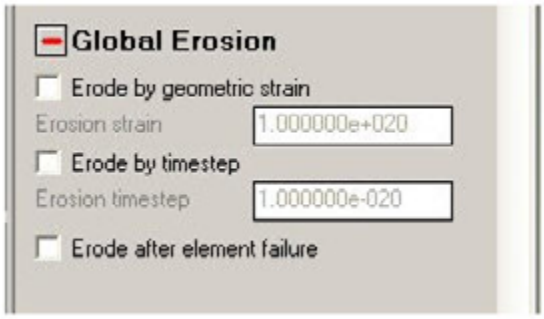
Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

When material is being transported out of an element that contains multiple materials or voids, it is necessary to use special rules to determine the amount of each material being transported, because the only information available for the transport algorithm is the fraction of each material in the individual elements. In Autodyn the Simple Lime Interface Calculation (SLIC) method is used which uses a small number of stereotypes of mixed material element topologies. The SLIC algorithm is the default and works well for all multi-material Euler analysis.

– **Updated SLIC**

Isolated small parts of material smaller than an element do not move through the mesh at the correct velocity. They tend to stop moving while surrounding material rushes by. The Updated SLIC option will transport these small material fractions with statistically the correct velocity. This option should be used with care, because it can have some undesirable side effects. It is possible for these small material fragments to "tunnel" through other multi-material Euler objects like walls or plates and emerge at the other side.

Global Erosion



These options define global erosion criteria for all elements in the model. The global erosion options are only available for unstructured elements.

20.1. Controls - Mass Scaling

Mass scaling is an artificial (numerical) mechanism for increasing the CFL (Courant-Friedrichs-Levy) timestep of individual elements that govern the maximum allowed timestep of explicit transient dynamic solutions in Autodyn. Increasing the timestep has the obvious benefit of reducing the number of cycles required to run a simulation to a given point in time. Educated use of this option can therefore result in significant improvements in efficiency.

The primary use of mass scaling is to increase the CFL timestep of a small number of very small (or stiff) elements in the model, which otherwise would have controlled the timestep for all elements. Using this option, Autodyn will automatically add artificial mass to individual elements to ensure that their CFL timestep is at least equal to a value defined by you. Mass scaling should only be used if the increased inertia of the scaled elements does not significantly affect the results of the simulation.

Mass scaling can also be used to increase the CFL timestep of elements that become highly distorted during the simulation. Extreme care should be taken to ensure that results obtained remain physical.

Mass scaling is activated under **Controls, Timestep** and the input parameters are defined as follows:

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Automatic Mass Scaling

Activate automatic mass scaling.

Frequency (cycles)

Cycle frequency that Autodyn will adjust the element mass scaling (this is not supported in parallel solutions). Recommended default is only at start-up (cycle 0). Post cycle 0 adjustment will only take place for solid elements (excluding ANP and NBS tetrahedra).

Min. CFL timestep

Mass scaling is applied to all elements with a CFL timestep less than the specified value.

Max. element scaling

Maximum mass scaling factor that can be applied to an element. Default is 100 (equating to 100 times original mass). If this limit is exceeded, no more mass is added to this element.

Max. part scaling

Maximum added mass that is allowed for an Autodyn Part, as a fraction of the original Part mass. Default is 0.05 (equating to 5% of the original mass. If this value is exceeded, the simulation will terminate with an error message.

Two new element variables have been introduced with the development. These variables are automatically activated when mass scaling is selected:

- TIMESTEP

  The CFL timestep for the element

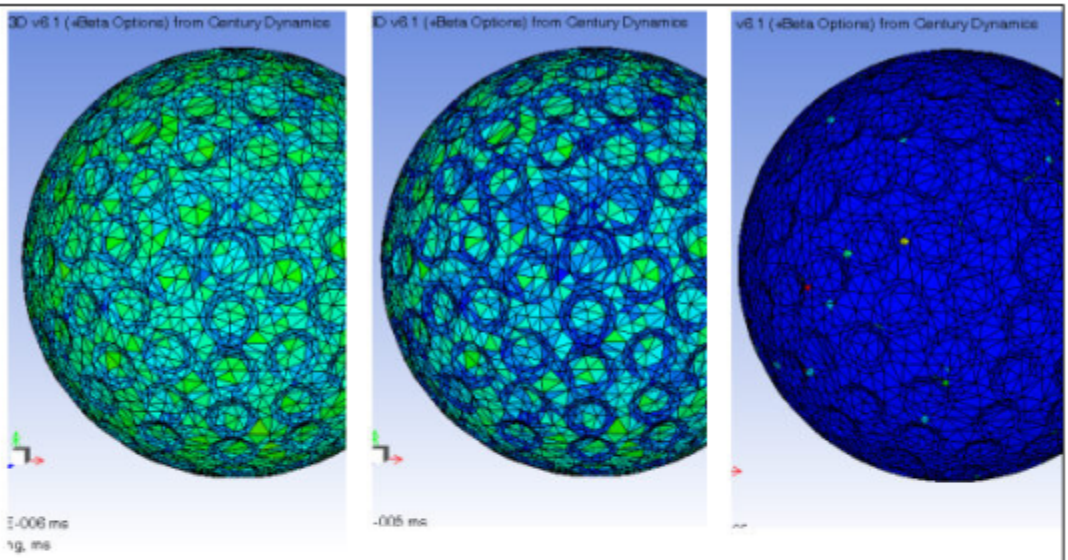
- MASS.SCALE

  The mass scale factor applied to the element. (1.0 indicates zero added mass, 2.0 indicates 100% added mass etc.)

A typical procedure to invoke mass scaling would be as follows

- Activate mass scaling retaining all the default parameters
- Run the simulation for 1 cycle
- Plot a contour of Timestep and review the element CFL timesteps. Identify a suitable timestep for mass scaling. The value chosen should exclude the majority of elements from mass scaling, and especially those in regions critical to the simulation results
- Re-load cycle zero
- Enter the selected Min CFL Timestep under Controls, Timestep
- Run the simulation

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Example application of mass scaling

Important notes and limitations:

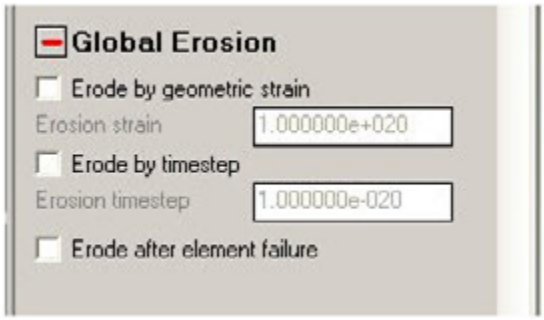
- Mass scaling will only be applied to Unstructured Elements/Parts
- Mass scaling is only applied to elements filled with materials which use a Linear or Hyperelastic equation of state
- The actual timestep used in a simulation will generally be lower than the Min CFL timestep you define
  - A safety factor is applied to get the actual timestep (default = 2/3)
  - The timestep used at cycle 1 of the simulation is additionally reduced by a factor of 2. The timestep is then allowed to increase by a maximum of 10% per cycle.
  - Large velocities can affect the timestep at cycle 0
  - Other features of the code will influence the timestep, such as artificial viscosity, blending, Gap contact.
- For Beams and Tet ANP elements, the minimum CFL timestep will be increased by using mass scaling. However, due to nodal averaging of some quantities that influence the timestep, the actual user defined CFL timestep may not be achieved.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- A summary of the contribution of mass scaling applied to the model can be obtained through
  - Plotting a contour of variable Mass Scale. This is a ratio of current mass to original mass (with scaling).
  - Review the `.prt` file output activated using **Output, Print, Energy Summary**

20.2. Controls - Global Erosion

A new global erosion option has been implemented that can be used to define global erosion criteria for all elements in the model. The global erosion options are only available for unstructured elements.



Three erosion criteria have been implemented and can be used in combination in an analysis:

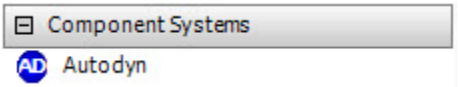
- Global Erosion by Geometric Strain
  - You can define a single erosion strain that will be used for all elements in the model.
    → The minimum of the global erosion strain, and the local material erosion strain (if specified) will be used to check for element erosion.
- Global Erosion by Timestep
  - You can define a single erosion timestep that will be used for all elements in the model.
    → The maximum of the global and the local material based erosion timestep (if specified) will be used to check for element erosion.
- Global Erosion by Failure
  - You can turn on erosion by material failure for all materials.
    → Using erosion by timestep will activate the element timestep variable which can be used in postprocessing.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Chapter 21: Autodyn Usage in Ansys

Accessing Autodyn

The Autodyn application is available in the Component Systems section of the Workbench Toolbox.



Use this application to access the full functionality of the Autodyn product. Please refer to the online Workbench documentation for an explanation of the Autodyn system components and supported operations.

The structural FE components of the Autodyn solver can also be accessed using the Mechanical Explicit Dynamics system located in the Analysis Systems section of the Workbench Toolbox. Please refer to the Explicit Dynamics Analysis Guide for further details.

Steps to Using Autodyn

There are three different approaches to using Autodyn within Workbench:

- Insert an Explicit Dynamics system from the Analysis Systems Toolbox. Use the Engineering Data, Geometry, Model and Setup cells to define the structural/mechanical model. Drag and drop an Autodyn component system onto the Setup cell of the Explicit Dynamics system. Double-clicking the Autodyn system Setup cell will start Autodyn and transfer the FE model definition (including materials, mesh, connections, loading, constraints, analysis settings) from the Explicit Dynamics system.
  - Can only set-up models for the FE solvers (including connections and contact)
  - 4-noded Tetrahedral meshes, 8-Noded Hexahedral Meshes, 4- and 3-noded Quad meshes, 2-noded Beam meshes
  - See Autodyn Walkthrough Using Explicit Dynamics (p. 512) for more details of the procedure.
- Insert a Mesh system from the Component Systems Toolbox. Use the Geometry and Mesh cells to define the structural/mechanical mesh. Drag and drop an Autodyn component system onto the Mesh cell of the Mesh system. Double-clicking the Autodyn system Setup cell will start Autodyn and transfer the FE model definition (including mesh and named selections) from the Mesh system to the Autodyn system.
  - Can only set up meshes for the FE solvers.
  - 4-noded Tetrahedral meshes, 8-Noded Hexahedral Meshes, 4- and 3-noded Quad meshes, 2-noded Beam meshes
  - See Autodyn Walkthrough Using Meshing (p. 519) for more details of the procedure.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

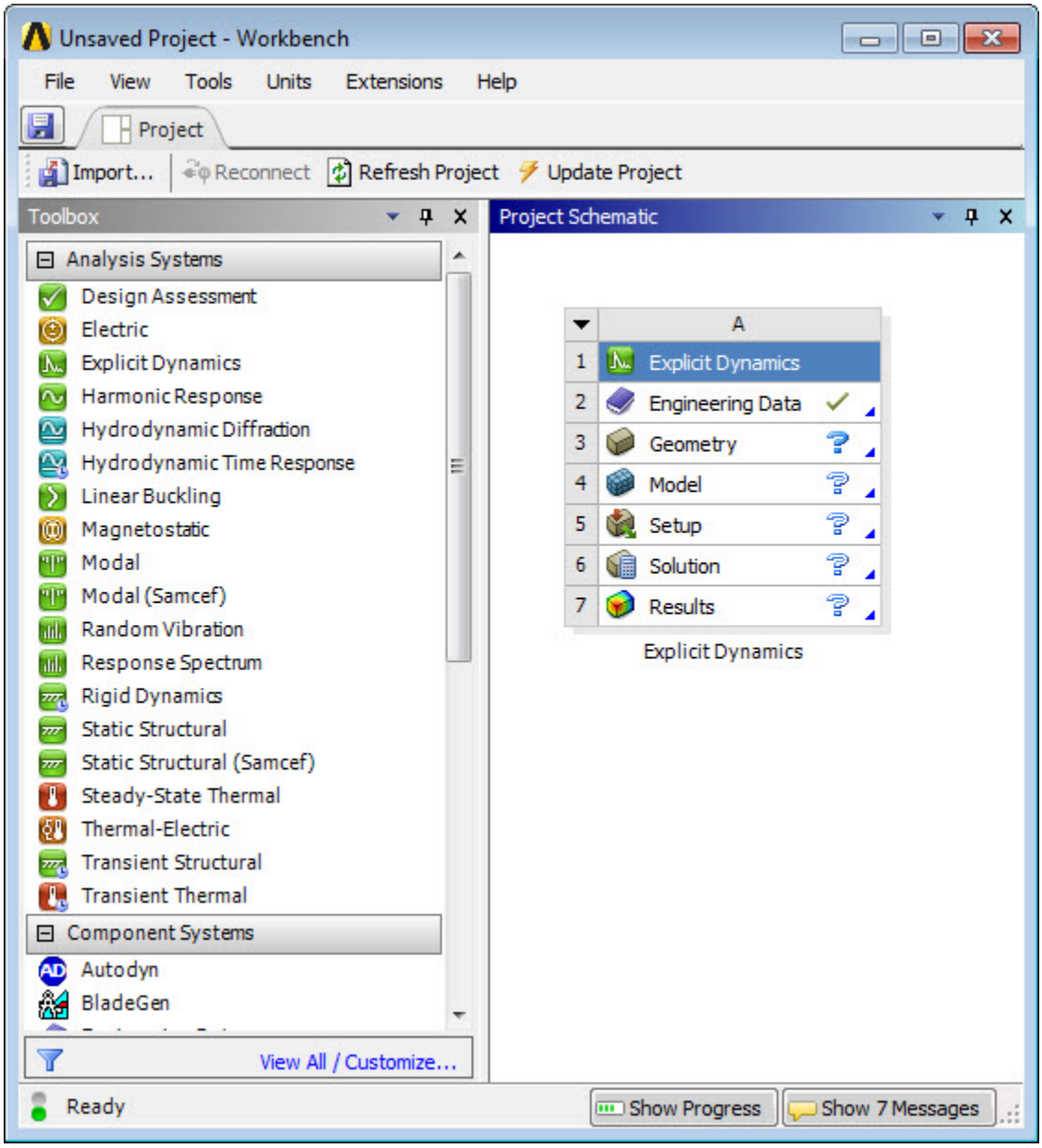
- For simple geometries, integrated geometry creation, and meshing tools within Autodyn may be used. Insert an Autodyn Component System and double-click the Setup cell to run and Autodyn application standalone.

  – Can set up models for all solvers.

21.1. Autodyn Walkthrough Using Explicit Dynamics

1. Start Workbench.

2. Insert an **Explicit Dynamics** system from the **Analysis Systems** toolbox.

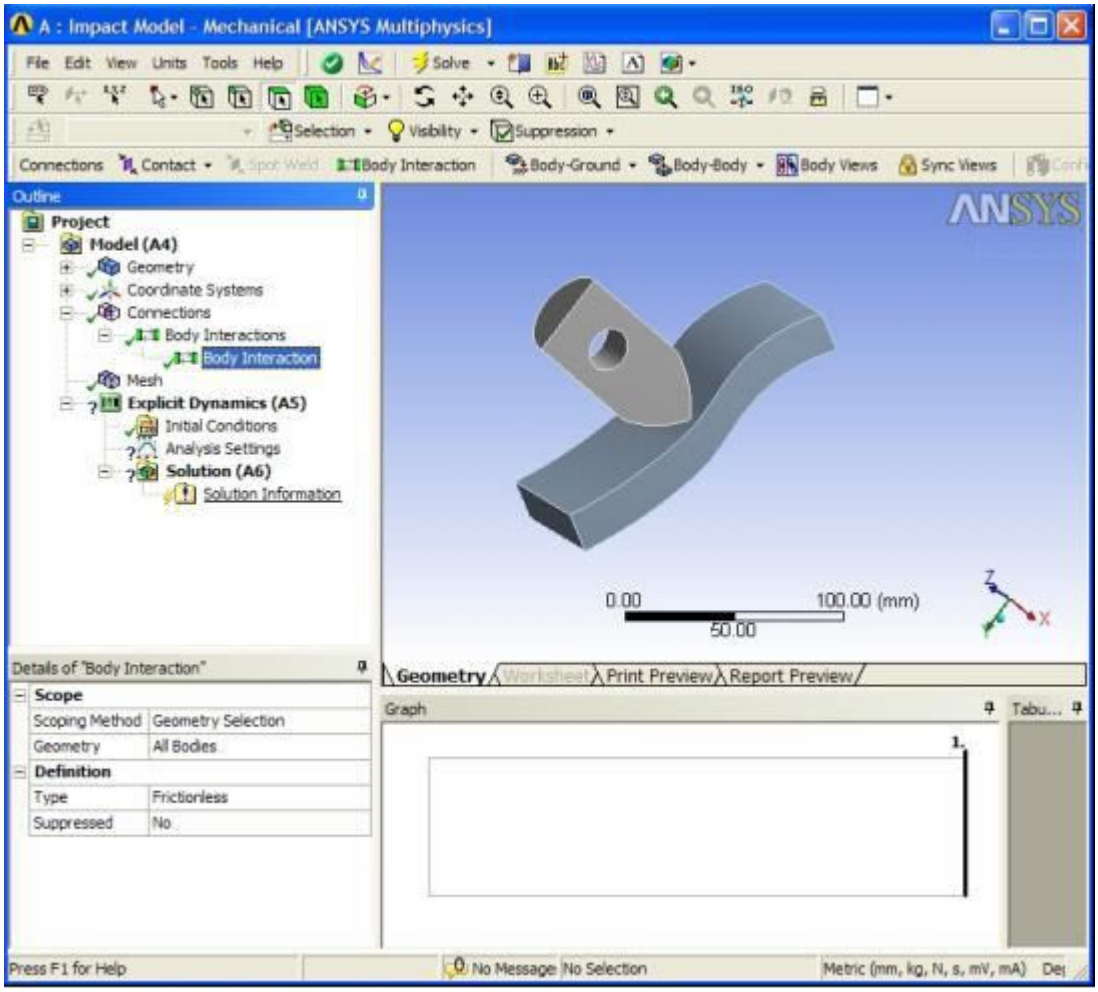


3. Right-click the Geometry cell (A3), and select **Import a Geometry**.

4. Double-click the Model cell (A4) to open up the Mechanical application editor.

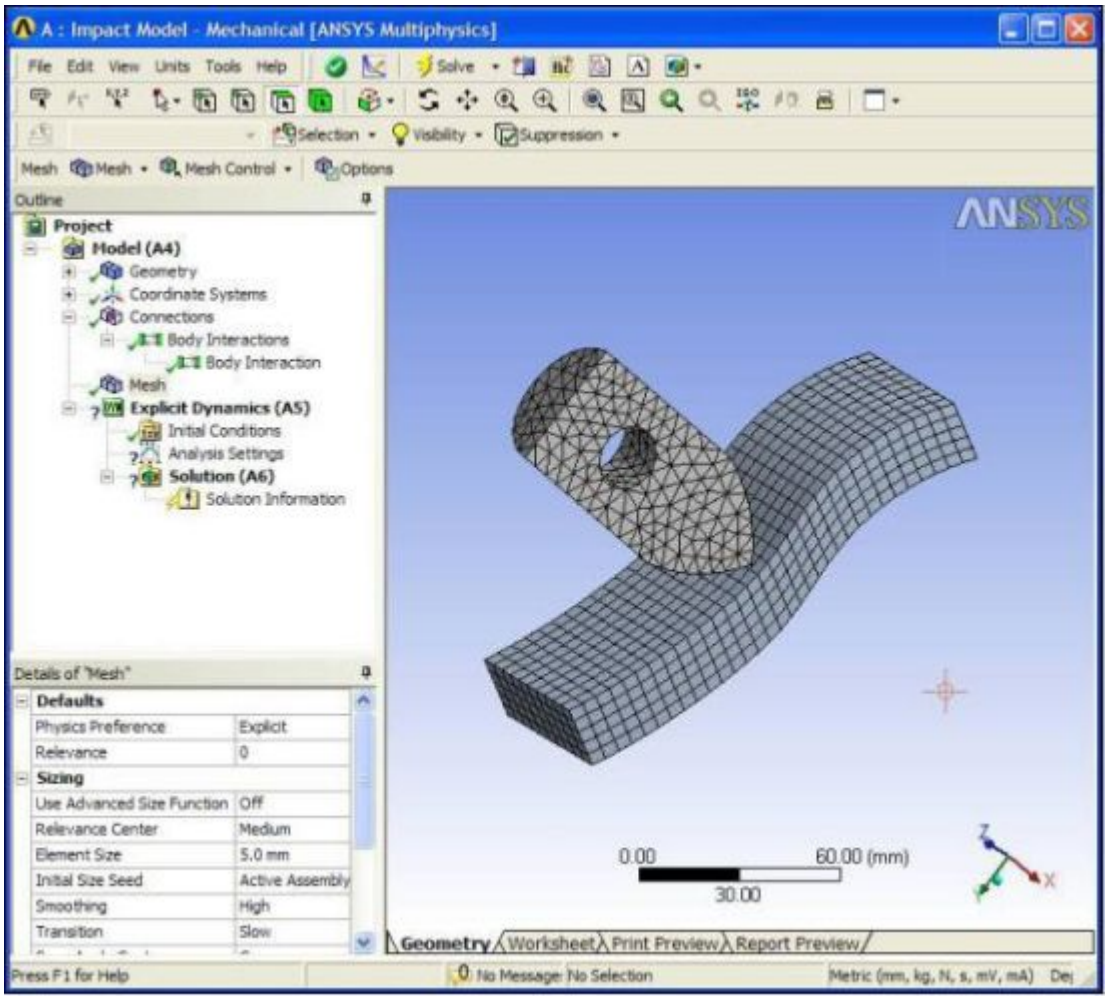
Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- Define the **Body Interactions**.



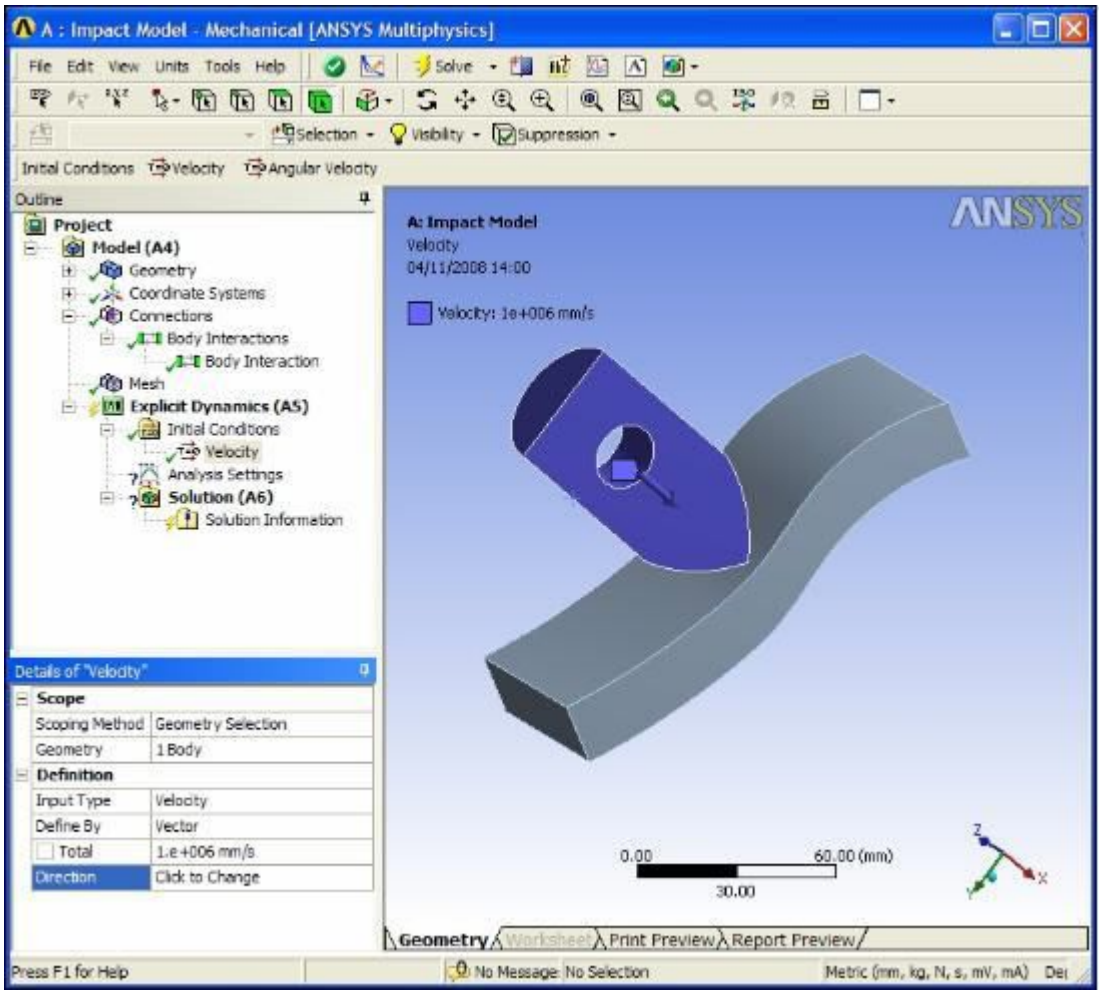
- Define the Mesh.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



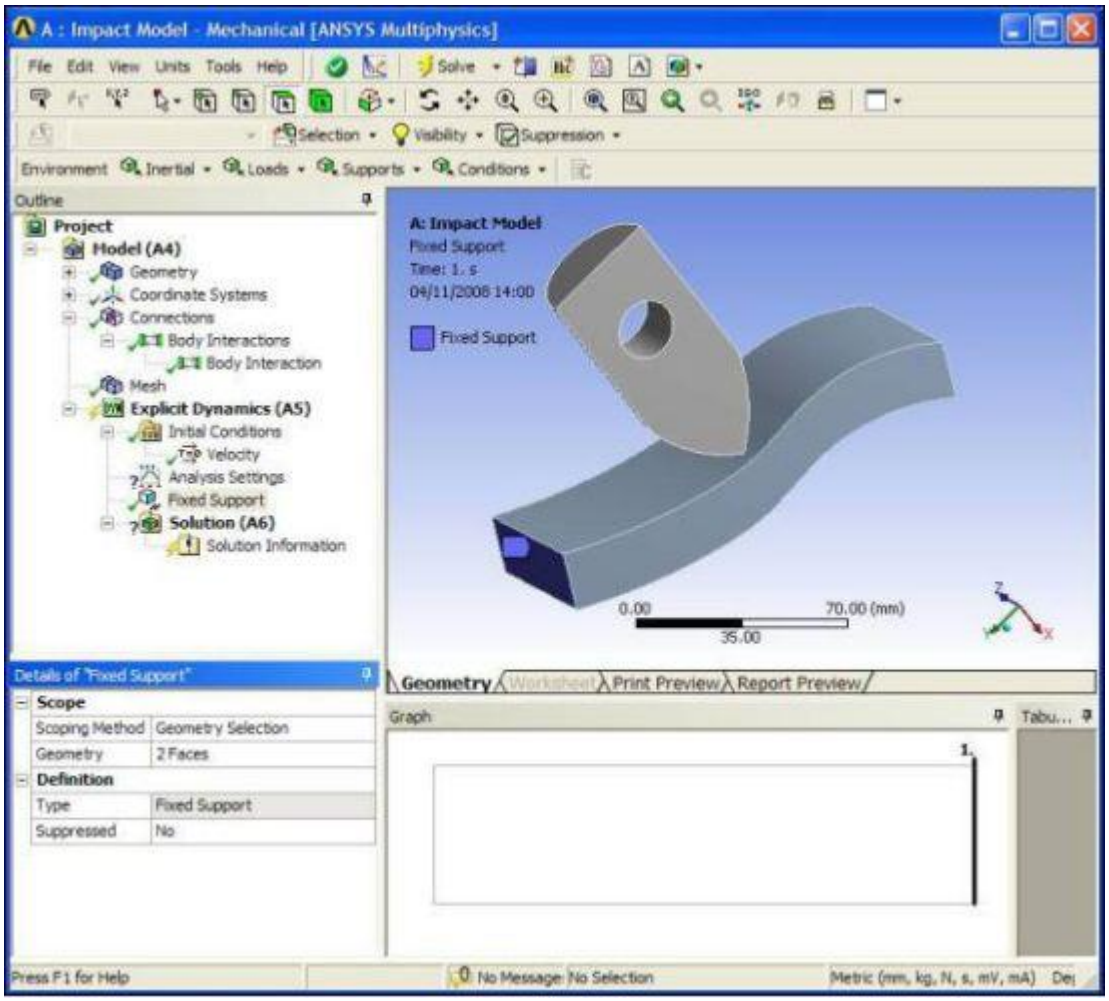
- Define Initial Conditions.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



- Define Boundary Conditions.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



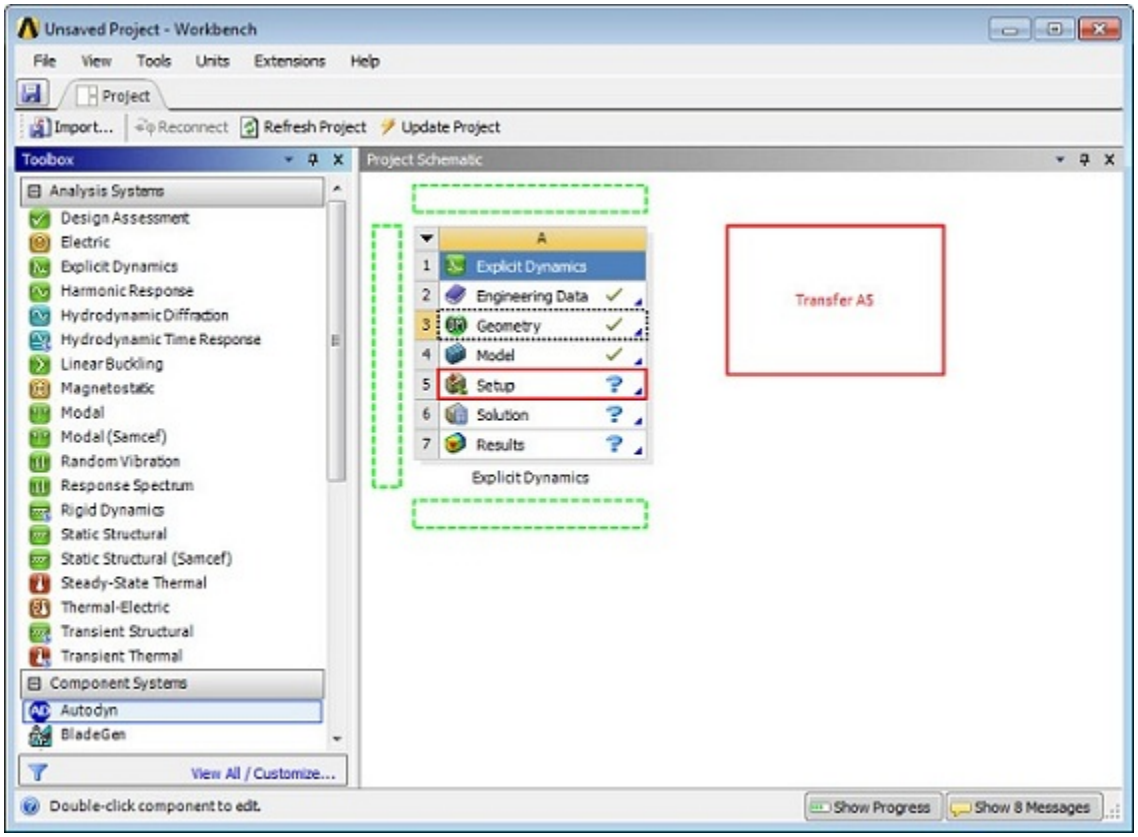
- Define Analysis Settings.

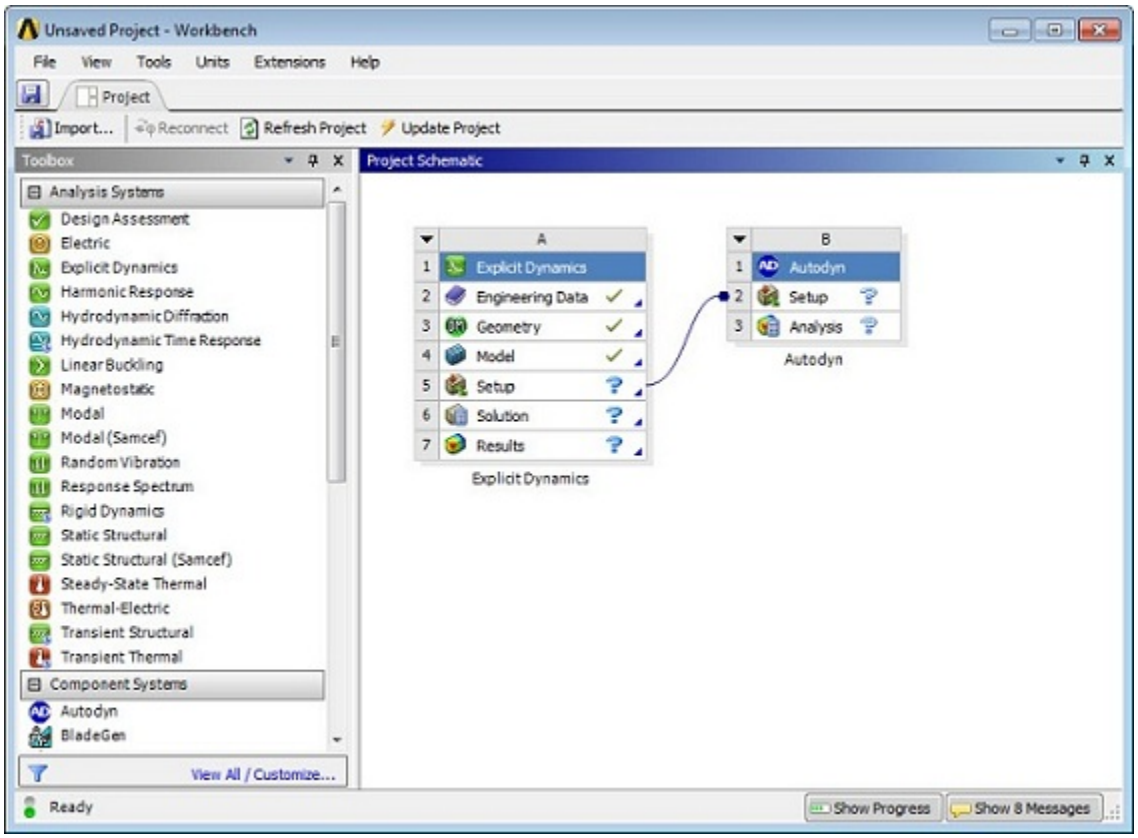
Details of "Analysis Settings"

Step Controls	
Resume From Cycle	0
Maximum Number of Cycles	1e+07
End Time	1.e-003 s
Maximum Energy Error	0.1
Reference Energy Time Step	0
Initial Time Step	Program Controlled
Minimum Time Step	Program Controlled
Maximum Time Step	Program Controlled
Time Step Safety Factor	0.9
Automatic Mass Scaling	No
Solver Controls	
Damping Controls	
Erosion Controls	
Output Controls	
Analysis Data Management	
Solver Files Directory	C:\Documents and Settings\raclegg\L...

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

5. Go to the Project Schematic and drag an Autodyn Component System onto cell A5 to generate the link between the Setup cell (A5) of the **Explicit Dynamics** system and the Setup cell (B2) of the Autodyn Component System.





Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

6. Transfer the **Explicit Dynamics** Setup to Autodyn.

   - Right-click **Explicit Dynamics** system Setup cell (A5), Select Update.
   - The model definition in the **Explicit Dynamics** system will be transferred into the Autodyn application.

     Double-click the Setup cell (B2) of the Autodyn system to open the Autodyn application.

     – Materials

     – Mesh

     – Initial Conditions

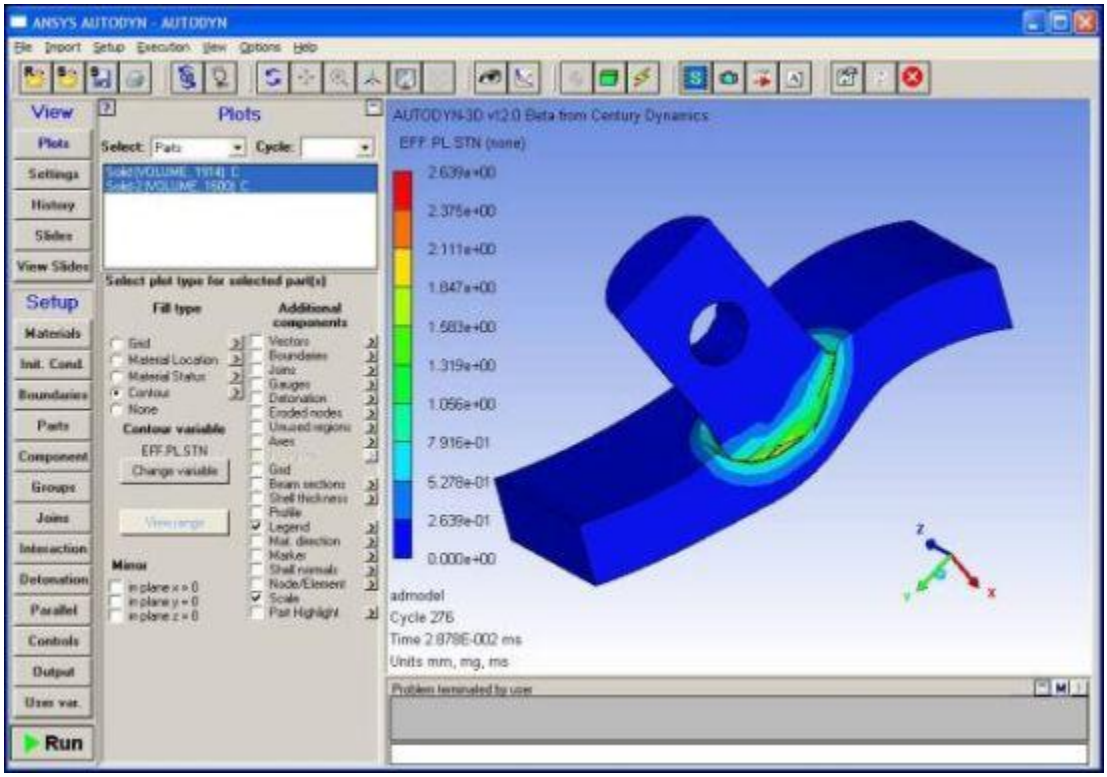
     – Boundary Conditions

     – Interactions

     – Controls

     – Output

   - Complete the model set up in Autodyn.
   - Execute the Simulation.
   - Postprocess results.



Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Explicit Dynamics-Autodyn Data Transfer

From this version, selecting **Update** or **Refresh** on an Autodyn system that is linked to an Explicit Dynamics system will result in all the existing data being overwritten by the latest data from the Explicit Dynamics system. This new behavior means that any intermediate modifications made in Autodyn will not be preserved after the update, i.e. the data from Autodyn and Explicit Dynamics is no longer merged.

To continue modifying the Autodyn system and preserve those changes, delete the link between the systems once all the desired data has been transferred from Explicit Dynamics to Autodyn.

Note:

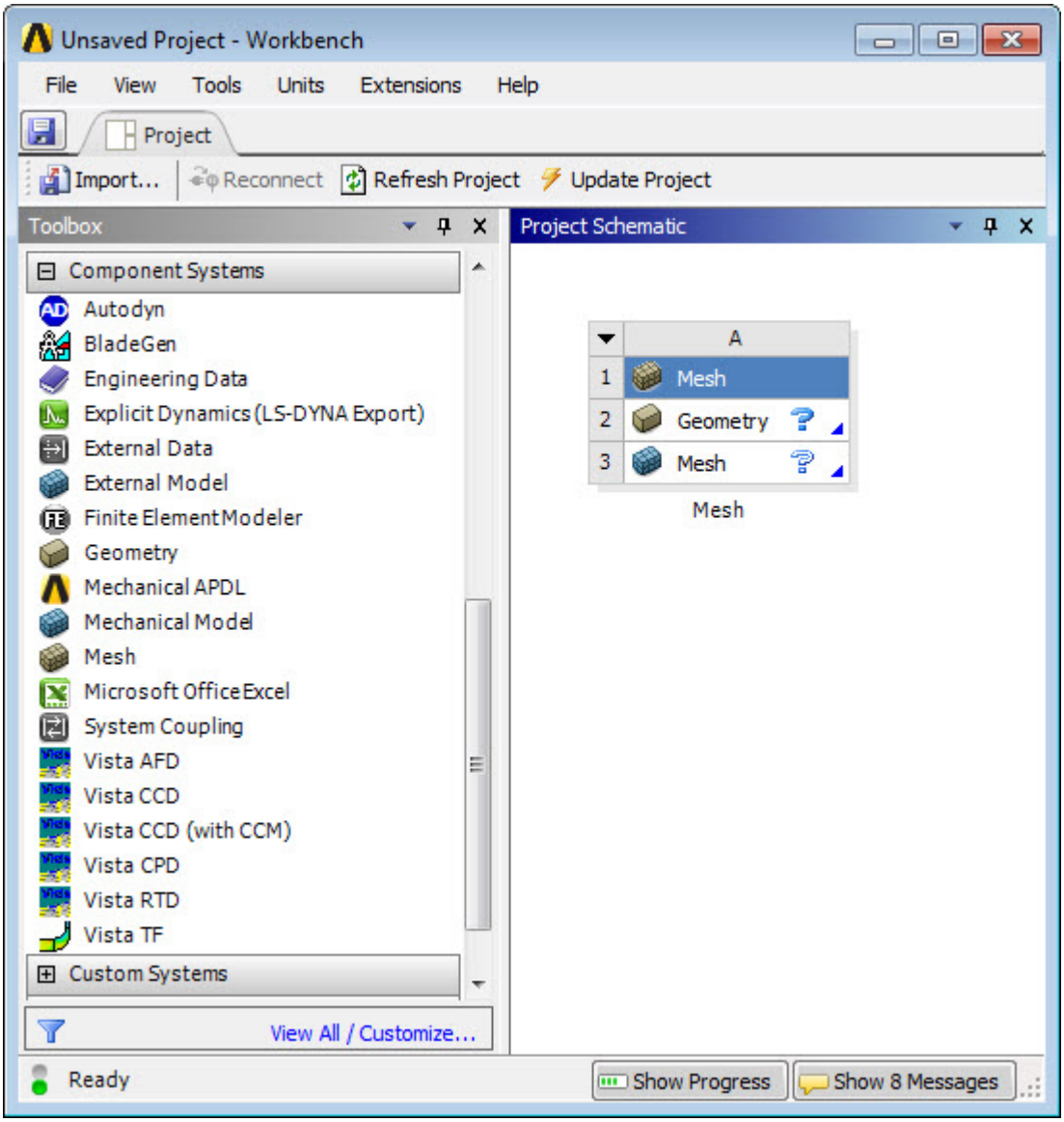
When running Workbench in scripted or background mode, be aware that pop-up messages about this new behavior must be blocked to successfully run the program. To block them, activate the **Autodyn.NoPrompts** flag either by running Workbench from the command line `C:\Program Files\ANSYS Inc\v232\Framework\bin\Win64\RunWB2.exe" -k Autodyn.NoPrompts` or by adding the value **Autodyn.NoPrompts** to the ANSYS_DEVELOPMENT_FEATURES environment variable. This ensures no pop-up messages are generated, and the Update or Refresh action proceeds.

21.2. Autodyn Walkthrough Using Meshing

1. Start Workbench.

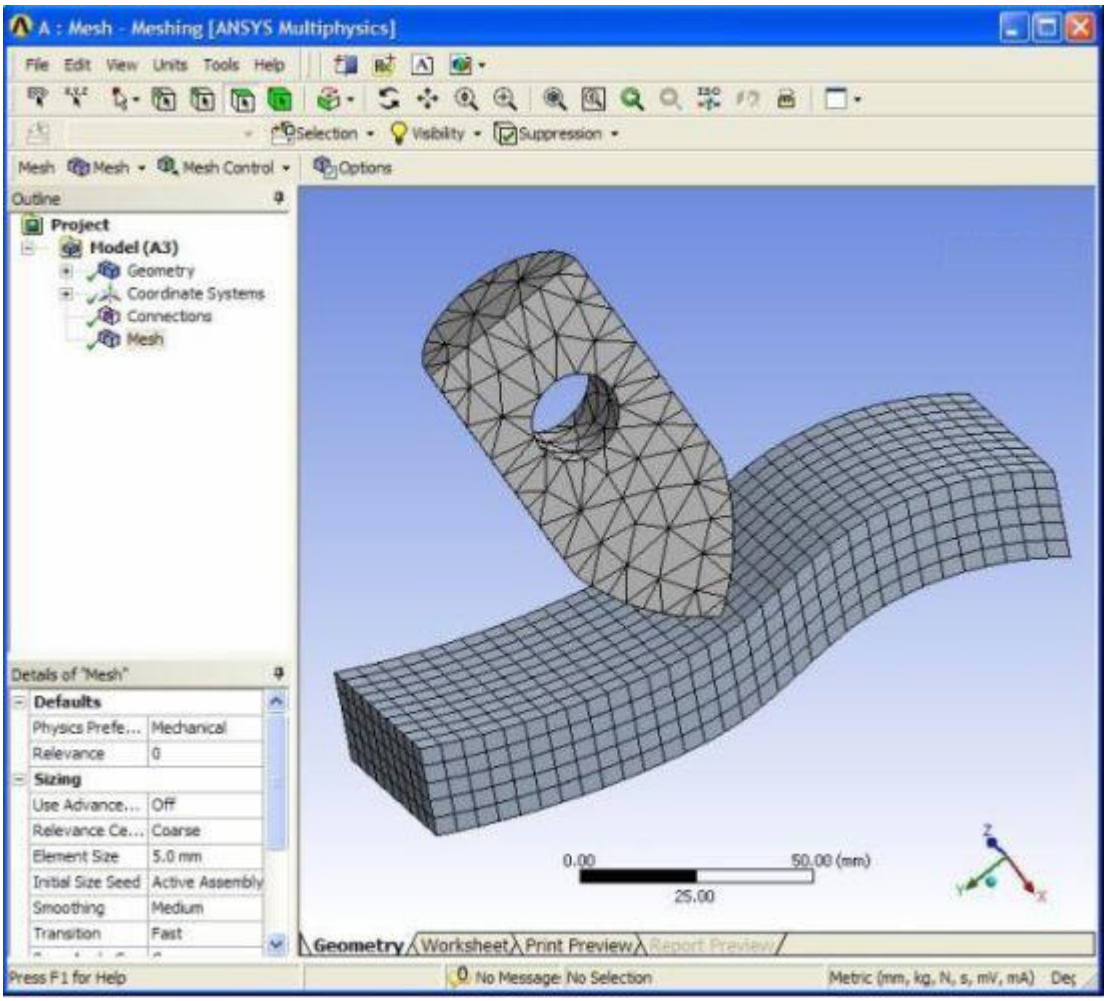
2. Insert a Mesh system from the **Component Systems** toolbox.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



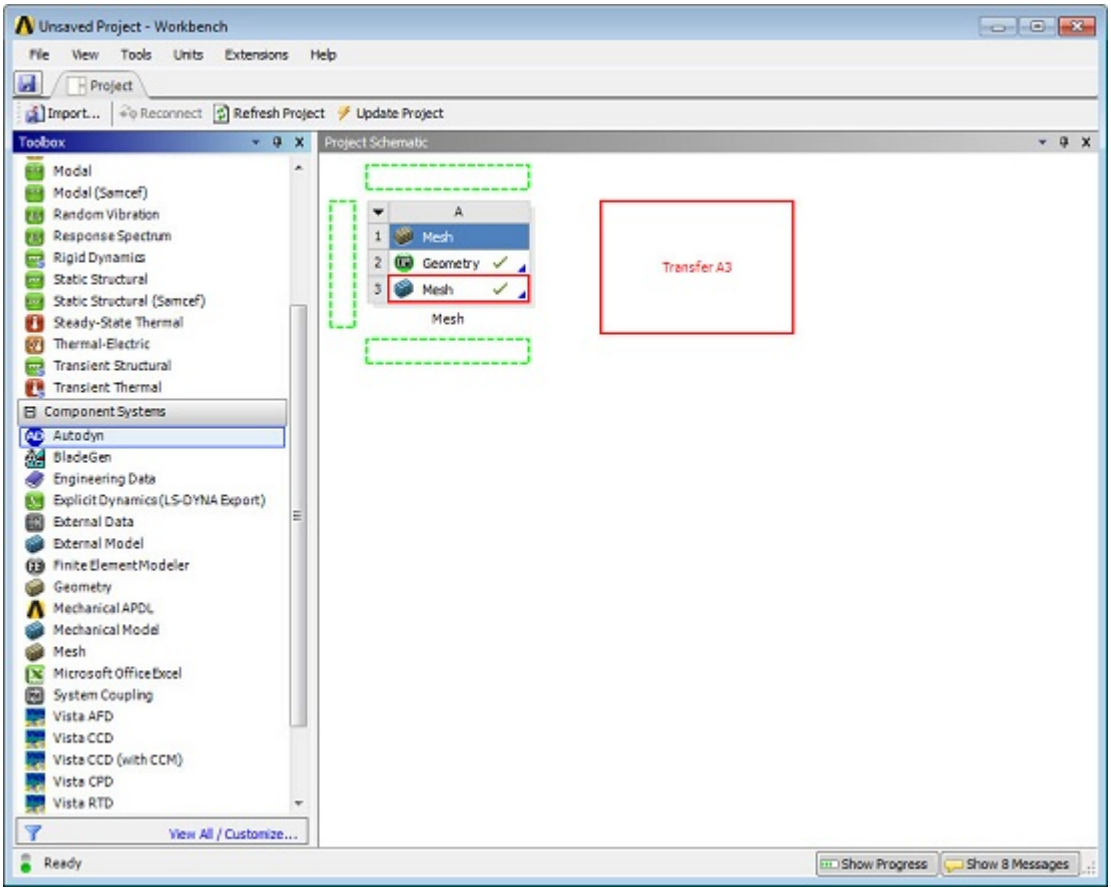
3. Right-click the **Geometry** cell (A2), **Import a Geometry**.
4. Double-click the **Mesh** cell (A3), to open up the Meshing application editor.
5. Define the Mesh.

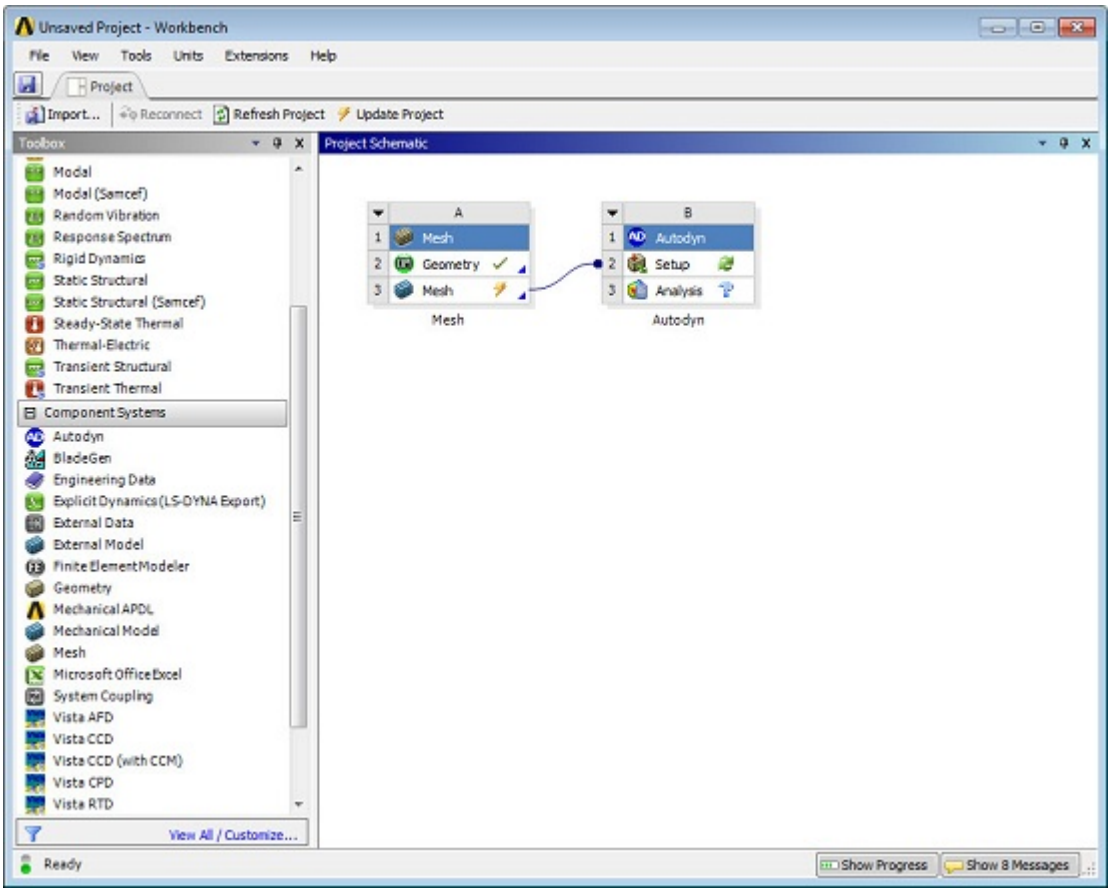
Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



6. Go to the **Project Schematic**, drag an Autodyn Component System onto cell A3, to generate the link between the Mesh cell (A3) of the Mesh system and the Setup cell (B2) of the Autodyn Component System.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

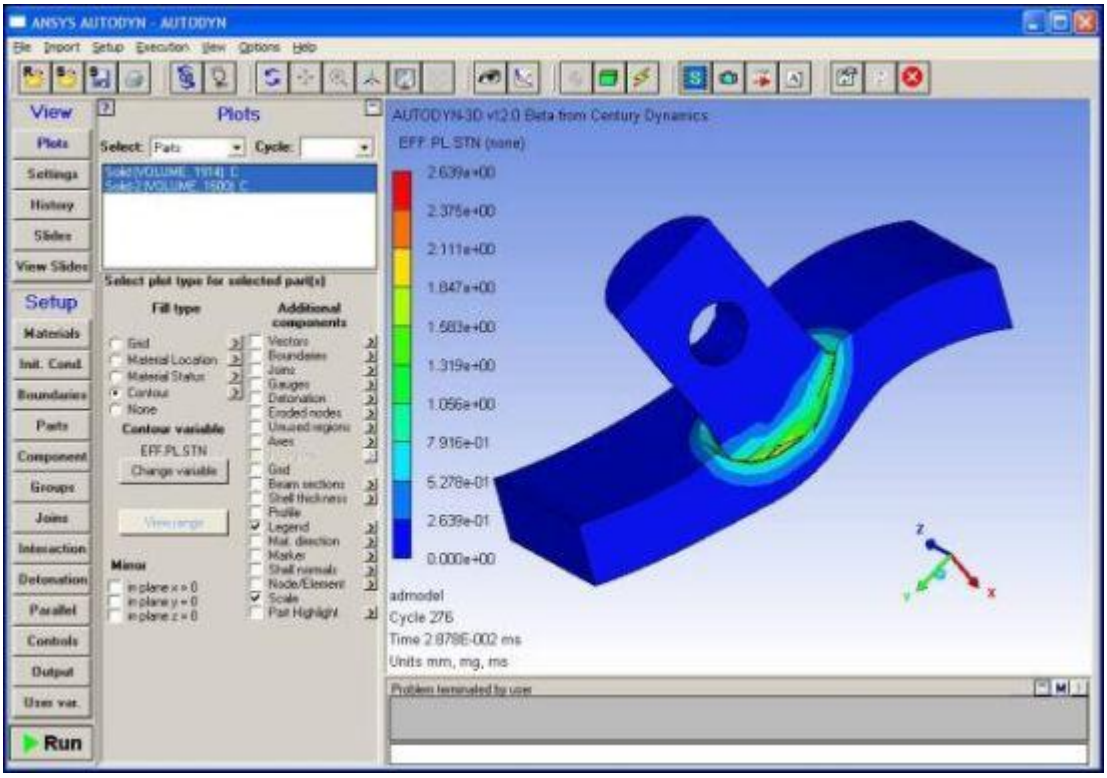




Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

7. Double-click the Setup cell (B2) of the Autodyn system to open the Autodyn application.

   - The model definition in the Mesh system will be transferred into the Autodyn application Mesh Named Selections.
   - Complete the model set up in Autodyn.
   - Execute the Simulation.
   - Post-process results.



Note:

Changes to material assignment, loads and initial conditions made in Autodyn will be lost after an update of geometry or mesh. It is recommended that you link the Autodyn system to an Explicit Dynamics system rather than to a Meshing system. Materials, loads, and so on may then be applied in the Explicit Dynamics system and will be retained after a geometry or mesh update.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information
of ANSYS, Inc. and its subsidiaries and affiliates.

Chapter 22: Autodyn Additional Information

The Autodyn solver is invoked when solving an Explicit Dynamics analysis system as part of a Mechanical application analysis, or solving an Autodyn Component system analysis in Workbench. Specific information is provided for material models used in the Autodyn solver. In addition, theoretical discussion of the Autodyn explicit dynamics analysis algorithms is provided.

22.1. Running Autodyn on Windows Platforms

22.2. Running Autodyn in Batch Mode on Linux Platforms

22.3. Files Created by Autodyn

22.4. Explicit Dynamics Material Models

22.5. Explicit Dynamics Theory

22.6. Autodyn Material Models

22.7. References

22.1. Running Autodyn on Windows Platforms

Autodyn, by default, runs in interactive mode, with the graphics window and interface displayed. If desired, you may also run Autodyn in batch mode where no graphics interface is shown. You may experience an increase in speed since the graphics do not need to be generated or refreshed. Also, by running Autodyn in batch mode it is possible to stack jobs to run consecutively while your PC is unattended (for example, overnight).

Autodyn can be executed from the operating system command line and accepts a number of command line arguments. The following command can be used to run Autodyn from the command line:

```
<installation path>\v251\aisol\AUTODYN\Winx64\AUTODYNWRAPPER.exe -noinput
```

The following table describes the command line arguments that can be used to control Autodyn file operations and execution behavior at start-up.

Argument	Operation Type	Operation
-noinput	Interactive (Serial\Parallel)	Opens the Autodyn GUI in order to create\execute models manually. In order to run a parallel simulation using the GUI, the -noinput option should be used in combination with the -mpi and -np options described below.
-i "<*Autodyn model file*>"	Batch (Serial\Parallel)	<*Autodyn model file*> is the name of the model file (including the path, if necessary). Using this option will run Autodyn in batch mode.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Argument	Operation Type	Operation
-beta		When running in batch mode, the model will run without beta options by default. Specifying this option will enable the model to run with beta features on.
-np <*number of worker processes*>	Batch (Parallel)	<*number of worker processes*> must be an integer that indicates the number of worker processes that will be used for the simulation. This option will have no effect on models that do not have the parallel options turned on.
-mpi <*mpi type*>	Batch (Parallel)	<*mpi type*> must be either `intelmpi`, `intelmpi2018`, or `msmpi`. `intelmpi` will run with the default version of Intel MPI (currently 2021.11) which is provided in the Ansys Install. `intelmpi2018` will run with Intel MPI version 2018.3.210 which is also provided with the Ansys Install. `msmpi` runs with Microsoft MPI.
-machines "<*machine name*>"	Batch (Distributed Parallel)	<*machine name*> must be a string containing a list of node names (computer names) that will be used to run the worker executables, and the number of worker processes that will be run on each machine. The master executable is run on the local machine (which is called the head node) by default. The string should be constructed with a machine name followed by a colon followed by the number of workers to be run on the machine. Multiple node definitions should be separated by commas; for example **`"MachineA:2,MachineB:3"`**.

Note:

When running a model with the AUTODYNWRAPPER using Intel MPI, it is assumed that the MPI files used are those included with the Ansys installation. It is possible to specify a different location for the MPI files by setting the environment variable I_MPI_ROOT; for example:

I_MPI_ROOT = C:\Program Files (x86)\Intel\oneAPI\mpi\2021.9.0

If running using Intel MPI version 2018 or earlier, you must also specify the `–mpi intelmpi2018` option to the AUTODYNWRAPPER.exe.

When running in batch mode, it is possible to request the creation of a save file and to stop the solve process before the end of the run by placing data files in either of the following directories:

- **The configuration folder.** On Windows 10 systems this is: C:\Users\username\AppData\Roaming\Ansys\v251\AUTODYN.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- **The data folder.** This is where the model input and output files are located.

Putting a file named `endbat.dat` in either of these folders will cause a simulation to stop at the end of the current cycle. Once stopped, Autodyn will delete the `endbat.dat` file.

Putting a file named `savenow.dat` in either of these folders will force the simulation to write a restart file at the end of the current cycle. Once the restart file has been written out, Autodyn will delete the `savenow.dat` file and it must be re-created to write out a new restart file.

An example batch file below shows how to stop a job that is running in batch mode.

```
REM write out endbat.dat in data directory
cd /D "D:\Analysis Data"
echo null>endbat.dat
```

Autodyn will use the user temporary folder for temporary files and log file creation. This folder must have read/write permission for the user. The location of the folder for Windows 10 systems is `C:\Users\`*`username`*`\AppData\Local\Temp`.

22.2. Running Autodyn in Batch Mode on Linux Platforms

An Autodyn analysis can be started in batch mode from the system command line using either:

- The AUTODYNWRAPPER. This method is the easiest way to start a serial or parallel analysis, where command line arguments allow the full definition of the parallel environment.
- The autodyn script. This method allows you to start a serial or parallel analysis but needs an additional parallel configuration file (`parallel.cfg`) and the definition of the environment variable ANSYS_AUTODYN_MPI_TYPE to select a non-default MPI type to run.

22.2.1. Running Autodyn using the AUTODYNWRAPPER

Autodyn can be executed from the operating system command line by using the AUTODYNWRAPPER.

The following command can be used to run Autodyn from the command line, using the AUTODYNWRAPPER, which accepts a number of command line arguments:

```
<installation path>/v251/Tools/mono/Linux64/bin/mono <installation path>/v251/autodyn/bin/AUTODYNWRAPPER.exe -i .
```

The following table describes the command line arguments that can be used to control Autodyn file operations and execution behavior at start-up.

Argument	Operation Type	Operation
-i "<*Autodyn model file*>"	Batch (Serial\Parallel)	<*Autodyn model file*> is the name of the model file (including the path, if necessary). Using this option will run Autodyn in batch mode.
-beta		When running in batch mode, the model will run without beta options by default. Specifying

Argument	Operation Type	Operation
		this option will enable the model to run with beta features on.
-np <*number of worker processes*>	Batch (Parallel)	<*number of worker processes*> must be an integer that indicates the number of worker processes that will be used for the simulation. This option will have no effect on models that do not have the parallel options turned on.
-mpi <*mpi type*>	Batch (Parallel)	<*mpi type*> must be either `intelmpi`, `intelmpi2018`, or `openmpi`. `intelmpi` will run with the default version of Intel MPI (currently 2021.11) which is provided in the Ansys Install. `intelmpi2018` will run with Intel MPI version 2018.3.222 which is also provided with the Ansys Install. `openmpi` runs with Open MPI.
-machines "<*machine name*>"	Batch (Distributed Parallel)	<*machine name*> must be a string containing a list of node names (computer names) that will be used to run the worker executables, and the number of worker processes that will be run on each machine. The master executable is run on the local machine (which is called the head node) by default. The string should be constructed with a machine name followed by a colon followed by the number of workers to be run on the machine. Multiple node definitions should be separated by commas; for example **`"MachineA:2,MachineB:3"`**.

Note:

When running a model with the AUTODYNWRAPPER using MPI parallel, it is assumed that the MPI files used are those included with the Ansys installation. It is possible to specify a different location for the MPI files by setting an environment variable:

- For Intel MPI, the I_MPI_ROOT environment variable can be set; for example:

```
setenv I_MPI_ROOT /opt/intel/compilers_and_libraries_2019.3.199/linux/mpi/intel64
```

- For Open MPI, the OPAL_PREFIX environment variable can be set; for example:

```
setenv OPAL_PREFIX <install dir>/tp/MPI/OpenMPI/4.0.5/linx64/
```

When running in batch mode, it is possible to request the creation of a save file and to stop the solve process before the end of the run by placing data files in the data directory where the model input and output files are located.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Putting an empty file named `endbat.dat` in the data folder will cause a simulation to stop at the end of the current cycle. Once stopped, Autodyn will delete the `endbat.dat` file.

Putting an empty file named `savenow.dat` in the data folder will force the simulation to write a restart file at the end of the current cycle. Once the restart file has been written out, Autodyn will delete the `savenow.dat` file and it must be re-created to write out a new restart file.

22.2.2. Running Autodyn using the `autodyn` Script

Once Autodyn has been installed on a Linux system, a script called **`autodyn251`** is provided that sets up the required runtime environment variables. This script is located within the `<Installation directory>/v251/autodyn/bin` directory. It can be run by either adding this directory path to your system path environment variable, or by typing the full path to the script. The script must be run using bash. It may throw errors if run using other interpreters such as dash.

The arguments to the script control which model is executed:

`autodyn -I {ident-name} -C {cycle-number}`

The C(ycle) argument is not required, and will default to cycle zero if not present.

Putting a file named `endbat.dat` in the running directory will cause a simulation to stop at the end of the current cycle. Once stopped, Autodyn will delete the `endbat.dat` file.

Putting a file named `savenow.dat` in the running directory will force the simulation to write a restart file at the end of the current cycle. Once the restart file has been written out, Autodyn will delete the `savenow.dat` file and it must be re-created to write out a new restart file.

In order to run a parallel calculation, you should place the parallel configuration file (`parallel.cfg`) in the same directory as the model file. For further details regarding the setup and execution of parallel simulations, see the Autodyn Parallel Processing Guide.

Note:

The MPI software used in a parallel calculation can be specified using the ANSYS_AUTODYN_MPI_TYPE environment variable. The available options are: intelmpi (Intel) or openmpi (OpenMPI). The default option for Linux is Intel MPI.

To use OpenMPI, the ANSYS_AUTODYN_MPI_TYPE variable needs to be set:

```
setenv ANSYS_AUTODYN_MPI_TYPE openmpi
```

For Intel MPI 2018.3 and older versions on systems with ptrace restrictions, such as Ubuntu, you need to set variable I_MPI_SHM_LMT=shm. This fix is documented on the Intel Community Forum. Newer versions of Intel MPI may not need this extra argument.

22.3. Files Created by Autodyn

In Windows, Autodyn will use the user temporary folder for temporary files and log file creation. The local user must have read/write permission to this folder. The location of this folder in Windows 7 systems

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

is `C:\Users\username\AppData\Local\Temp`. On Linux, temporary session files and log files are written to the model data folder.

File Name	File Content
autodyn.log	Session log file that provides an overview of the models that have been run and possible error messages.
version.log	Autodyn session file that provides an overview of the licenses obtained from the license server.

Autodyn files associated with a particular analysis model with name `my_ident` are created in the model data folder.

File Name	File Content
my_ident_0.ad	Analysis model file containing the original model setup.
my_ident_xxx.ad	Save file with the full model data at cycle xxx. Analysis can be restarted or continued from this file.
my_ident.fil	Remap file containing the results of a previous analysis to be mapped onto the current model.
my_ident.his	Time history file containing the time histories of variables at specified gauge locations.
my_ident.log	Analysis model log file containing analysis run data such as cycle summaries and CPU usage data.
my_ident_workerxx.log	Parallel model log file from worker number xx which provides analysis data from the part of the model that resides on worker xx.
my_ident.prt	Analysis print out file which provides analysis model summaries and user requested print output.
my_ident_xxx.res	Results file with user selected model data at cycle xxx. Resides in sub-folder `my_ident_adres`. Analysis can not be restarted or continued from this file.
my_ident.sum	Summary file containing material and part summary data.
my_ident.uhs	User history file containing text output (HTML or CVS) of simulation time history plots.

22.4. Explicit Dynamics Material Models

A description of the material models used by the Autodyn solver in Explicit Dynamics analyses in Workbench can be found in Material Models Used in Explicit Dynamics Analysis in the *Explicit Dynamics Analysis Guide*.

22.5. Explicit Dynamics Theory

A description of the analysis theory used by the Autodyn solver in Explicit Dynamics analyses in Workbench can be found in Explicit Dynamics Theory Guide in the *Explicit Dynamics Analysis Guide*.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

22.6. Autodyn Material Models

Autodyn can use material models available within Explicit Dynamics in Workbench plus others only currently available in the Autodyn system. A description of the material models used by the Autodyn solver in Explicit Dynamics analyses in Workbench can be found in Material Models Used in Explicit Dynamics Analysis in the *Explicit Dynamics Analysis Guide*. Material Models specific to Autodyn are described in the following sections. These models can be selected when setting up or modifying a material from the Material New/Modify panel.

22.6.1. Equation of State

22.6.2. Material Strength

22.6.3. Failure Models

22.6.4. Material Cutoffs

22.6.1. Equation of State

A general material model requires equations that relate stress to deformation and internal energy (or temperature).

In most cases, the stress tensor may be separated into a uniform hydrostatic pressure (all three normal stresses equal) and a stress deviatoric tensor associated with the resistance of the material to shear distortion (as is the case for most materials in Autodyn).

Then the relation between the hydrostatic pressure, the local density (or specific volume) and local specific energy (or temperature) is known as an equation of state.

The equation of state can be determined from knowledge of the thermodynamic properties of the material and ideally should not require dynamic data to build up the relationship. However, in practice, the only practical way of obtaining data on the behavior of the material at high strain rates is to carry out well-characterized dynamic experiments. It is important to recognize that, since the relationship is required for use in a numerical code, an analytic form is chosen to facilitate solution. Such an analytic form is at best an approximation to the true relationship. Further, the equation of state may be given in extensive tabular form and in that case the analytic form chosen can be considered as an interpolation relationship.

This section describes the Equations of State available in Autodyn that are not described in the Equations of State in the *Explicit Dynamics Analysis Guide*.

22.6.1.1. Tillotson EOS

This form of equation of state (Tillotson 1962 [1], Allen 1967 [2]) was derived to provide an accurate description of the material behavior of metallic elements over the very wide ranges of pressure and density met in hypervelocity phenomena.

Not only must such an equation of state describe normal density material and its condition after shock, but also its expansion and change of phase in cases where the shock energy has been sufficient to melt or vaporize the material.

The pressure range can be so large that the "low pressure" regime of this form of equation of state is defined as from 0 to 10 Mbar and "high pressure" from 10 to about 1000 Mbar. Thus any pressure and results from normal laboratory experiments cover only the "low pressure" region.

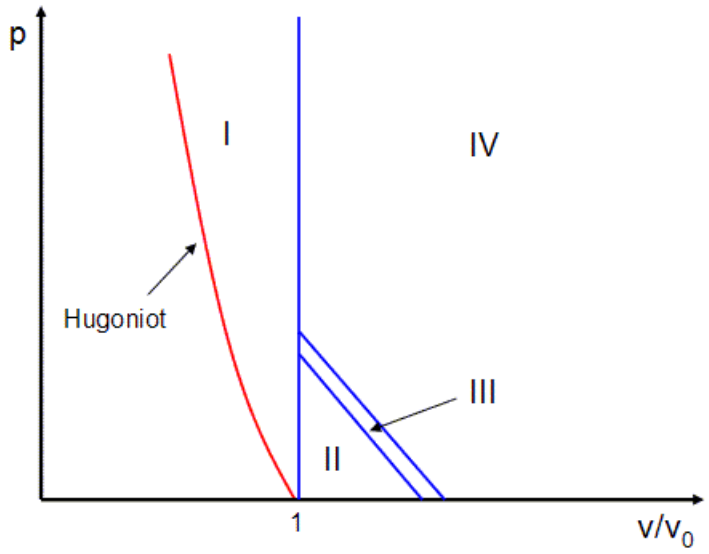
Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

For the derivation of an equation of state for the "high pressure" region, analytic forms provide best fit interpolations between Thomas-Fermi-Dirac data at high pressures (above 50 Mbar) and experimental data at low pressures.

The formulation is claimed to be accurate to within 5% of the Hugoniot pressure and to within 10% of the isentropic pressures. It is therefore a very useful form of equation of state to use for general hypervelocity impact problems.

22.6.1.1.1. Regions of Interest

The total range of the pressure-volume plane is divided into four regions as shown in the following figure.



The region to the left of the Hugoniot can only be reached by adiabatic (non-shock) compression and is not relevant to impact problems. It is therefore excluded from the formulation.

Region I represents the compressed phase of the material and extends vertically to shock pressures of about 1000 Mbar.

Region II describes material which has been shocked to an energy less than the sublimation energy e_s and will therefore, on adiabatic release, return to zero pressure as a solid.

There is no provision in this equation of state to describe the material at pressures less than zero.

Region IV is the expansion phase of material which has been shocked to an energy e_s' sufficiently large to ensure that it will expand as a gas at very large expansions.

For convenience the change of phase is assumed to take place at $v = v_0$.

For large specific volumes, the formulation for Region IV extrapolates to an ideal gas limit.

It is desirable, or even necessary, to ensure that the formulations in each region provide continuous values of the pressure and its first derivatives at the boundaries between contiguous regions.

This is true for the boundary between Regions I and II at $v = v_0$ but if e_s' is set equal to e_s difficulties may arise at the boundary between Regions II and IV at volumes slightly greater than v_0.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

To avoid this problem Region III is defined by setting

$$e_s'=e_s+ke_v \tag{22.1}$$

where e_v is the vaporization energy at zero pressure determined, like the boiling point energy e_s, from thermodynamic data.

Region III lies between Regions II and IV where $e_s < e < e_s'$. In this region the pressure is calculated as a mean between that calculated for Regions II and IV.

The constant k is determined empirically to ensure a well behaved solution and in practice satisfactory solutions have been obtained with values of k in the range of zero to one fifth.

22.6.1.1.2. Regional Formulations

In what follows let:

$$\begin{aligned}\eta&=\frac{\rho}{\rho_0}\\ \mu&=\eta-1\\ \omega_0&=1+\frac{e}{e_0\eta^2}\end{aligned} \tag{22.2}$$

You are prompted for the reference density ρ_{ref} (or ρ_0 in the equations below) and the constants in the Tillotson equation of state viz. a, b, A, B, α, β, e_0, e_s, and e_s'.

For Region I ($\mu \geq 0$) the pressure p_1 is given by a Mie-Gruneisen equation of state, but since the formulation is to be valid for a very large range of pressure, the Gruneisen Gamma is a function of both v and e, not just a function of v alone.

The constants fit the low pressure shock data but they are adjusted to fit the asymptotic Thomas-Fermi behavior for the variation of pressure at maximum compressions (like a monatomic gas).

The formulation for Region II is as for Region I with a slight modification to one term to avoid difficulties as μ goes increasingly negative.

In Region IV the formulation is chosen to give the correct behavior both at high pressure/normal density and for very large expansion (where it must converge to an ideal gas behavior).

With these constraints the different formulations are:

Region I ($\mu \geq 0$)

The pressure p_1 is given by

$$p_1=\left(a+\frac{b}{\omega_0}\right)\eta\rho_0 e+A\mu+B\mu^2 \tag{22.3}$$

Region II ($\mu < 0$, $e \leq e_s$)

The pressure p_2 is given by the same formulation as that for Region I except that B = 0, i.e.

$$p_2=\left(a+\frac{b}{\omega_0}\right)\eta\rho_0 e+A\mu \tag{22.4}$$

Region III ($\mu < 0$, $e_s < e < e_s'$)

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

The pressure p_3 is given by

$$p_3 = p_2 + \frac{(p_4 - p_2)(e - e_s)}{(e_s' - e_s)} \tag{22.5}$$

Region IV ($\mu < 0$, $e \geq e_s'$)

The pressure p_4 is given by

$$p_4 = a\eta\rho_0 e + \left(\frac{b\eta\rho_0 e}{\omega_0} + A\mu e^{\beta x}\right) e^{-ax^2} \tag{22.6}$$

where $x = 1 - 1 / \eta$.

22.6.1.2. Puff EOS

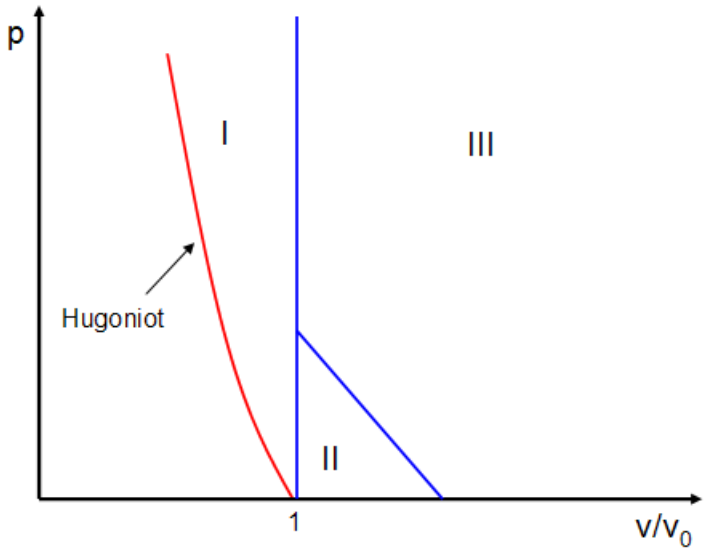
Like the Tillotson EOS, this equation of state was constructed to cover the behavior of material from cold shocked regions to hot, highly expanded regions.

The equation of state was first used in the PUFF/KO hydrocodes (Brodie & Hormuth [3], 1966: Bakken & Anderson, 1969 [4]) and is based on a Mie-Gruneisen form but with a variable Gruneisen Gamma in the expanded phase to give the required convergence to perfect gas behavior at very large expansions if the energy e ($\rho = \rho_0$) is greater than the sublimation energy e_s.

Like the Tillotson EOS, the (p, v) plane is divided into separate regions according to whether the material is compressed or expanded, and if expanded whether the energy is less than or greater than the sublimation energy.

22.6.1.2.1. Regions of Interest

The total range of the pressure-volume plane is divided into three regions as shown in the following figure.



The region to the left of the Hugoniot can only be reached by adiabatic (non-shock) compression and is not relevant to impact problems. It is therefore excluded from the formulation.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Region I represents the compressed phase of the material and extends vertically to shock pressures of about 1000 Mbar.

Region II describes material which has been shocked to an energy less than the sublimation energy e_s and will therefore, on adiabatic release, return to zero pressure as a solid.

There is no provision in this equation of state to describe the material at pressures less than zero.

Region III is the expansion phase of material which has been shocked to an energy e_s sufficiently large to ensure that it will expand as a gas at very large expansions.

22.6.1.2.2. Regional Formulations

You are prompted for a Gruneisen Gamma, Γ_0, an expansion coefficient, H (the effective ideal gas constant at large expansions, γ - 1) the sublimation energy, e_s, and constants A_1, A_2, A_3, T_1, and T_2.

Then the formulation is:

Region I ($\mu \geq 0$)

The pressure p_1 is given by

$$p_1=\left(A_1\mu+A_2\mu^2+A_3\mu^3\right)\left(1-\Gamma\mu/2\right)+\Gamma\rho e \tag{22.7}$$

in this equation $\Gamma\ \rho = \Gamma_0\ \rho_0$.

Region II ($\mu < 0$, $e < e_s$)

The pressure p_2 is given by

$$p_2=\left(T_1\mu+T_2\mu^2\right)\left(1-\Gamma\mu/2\right)+\Gamma\rho e \tag{22.8}$$

If T_1 is input as zero it is reset to $T_1 = A_1$

Region III ($\mu < 0$, $e \geq e_s$)

The pressure p_3 is given by

$$p_3=\rho\left[H+\left(\Gamma_0-H\right)\eta^{1/2}\right]\left[e-e_s\left\{1-\exp\left(N\left(\eta-1\right)/\eta^2\right)\right\}\right] \tag{22.9}$$

where $N = A_1 / \rho_0\ \Gamma\ e_s$.

Unlike the Tillotson EOS, there is no interpolative region between regions II and III.

Key Consideration for Puff EOS

It is important to note that the implementation of the Puff EOS in Autodyn differs from what is described in Bakken & Anderson [4]. The variable **ALPHA** is used to track the region out of the three described above. Typically, this is an integer (whole) number (see the comment below for Eulerian grids). It effectively determines which equation of state calculation is used as follows:

- If **ALPHA**<= 0.5, then use the **Region I** equation

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- If 0.5<**ALPHA**<1.5, then use the **Region II** equation
- If **ALPHA**>= 1.5, then use the **Region III** equation

ALPHA is calculated prior to the calculation of the equation of state and is set to:

- 0.0 if μ>= 0
- Either 1.0 or 2.0, if the internal energy is less than or more than the sublimation energy. However, note that **ALPHA** is only updated if **ALPHA** from the previous cycle was less than a half.

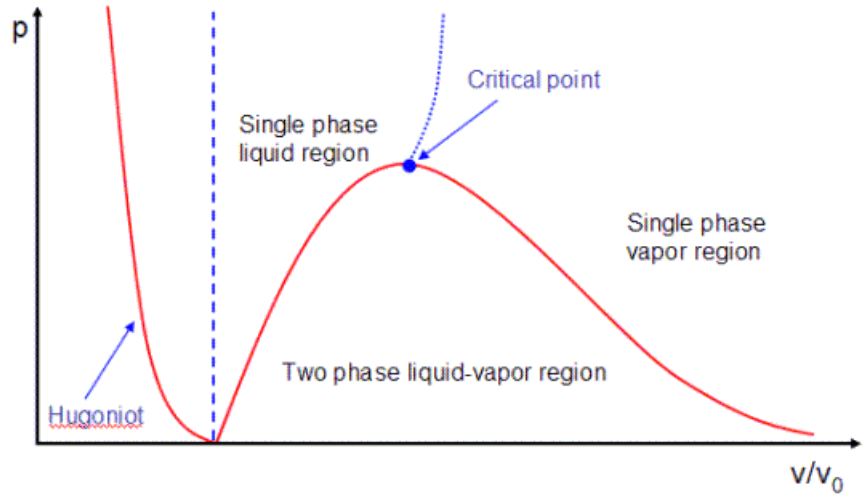
If none of the above conditions are met, then the value of **ALPHA** remains the same as the previous cycle.

An additional note for 3D Eulerian grids is that the value of **ALPHA** is transported in the advection step of the Eulerian cycle which may result in non-integer values of **ALPHA** when plotted or examined in the Autodyn GUI.

22.6.1.3. Two-Phase EOS

This formulation (Morgan 1973 [5]) only considers material which has expanded from normal density but gives a realistic description of both the single phase (liquid or vapor) and the two-phase (in which liquid and vapor co-exist) behavior of the expanding material, provided that the conditions along the saturation curve for this material can be tabulated.

The pressure-volume (p, v) plane for $v > v_0$ is divided into two regions by the saturation curve as shown in the following figure.



Above the saturation curve the material exists in a single phase state, either liquid or vapor, while below the saturation curve both phases coexist.

On the saturation curve the values of p, v, e, Γ, and temperature T are tabulated at suitable intervals to ensure accurate interpolations in the required calculations.

This equation of state should be used with some other form which will cover the region of compressed states (e.g. Shock or Polynomial). In this way the whole (p, v) plane will be covered.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

22.6.1.3.1. Single Phase Region

In the single phase region the behavior of the material is described by a Gruneisen equation of state of the form

$$p=p_r(v)+\frac{\Gamma(v)}{v}\left(e-e_r(v)\right) \tag{22.10}$$

where in this case p_r (v), e_r (v) and Γ_r (v) are the tabulated functions along the saturation curve and are determined for a specific value of v by interpolation within the tabular entries.

It is assumed that on the liquid side of the saturation curve T_r, e_r and Γ_r vary quadratically with v_r along the curve and that p_r may be obtained from a relationship of the form

$$\log p_r=\frac{A_1}{T_r}+B_1 \tag{22.11}$$

On the vapor side of the saturation curve e_r and Γ_r are taken to vary linearly with T_r, and v_r and T_r are assumed to be related by

$$\log v_r=\frac{A_2}{T_r}+B_2 \tag{22.12}$$

and

$$\log p_r=\frac{A_1}{T_r}+B_1 \tag{22.13}$$

22.6.1.3.2. Two-Phase Region

In the two-phase region below the saturation curve the specific volume v of the liquid-vapor mixture is written as

$$v=\alpha v_g+\left(1-\alpha\right)v_1 \tag{22.14}$$

and the internal energy e as

$$e=\alpha e_g+\left(1-\alpha\right)e_1 \tag{22.15}$$

where α is the mass fraction of vapor in the liquid-vapor mixture and v_g, v_l, e_g and e_l are the specific volumes and internal energies of the saturated liquid and vapor at the saturated vapor pressure p and temperature T, i.e. they are conditions on the saturation curve.

Eliminating α from the two equations above gives

$$\frac{\left(v-v_g\right)}{\left(v_g-v_1\right)}=\frac{\left(e-e_g\right)}{\left(e_g-e_1\right)} \tag{22.16}$$

This is an implicit equation defining p (or T) in terms of v and e.

Data on the saturation curve for water has been extracted from Morgan's paper and included as a standard option in Autodyn.

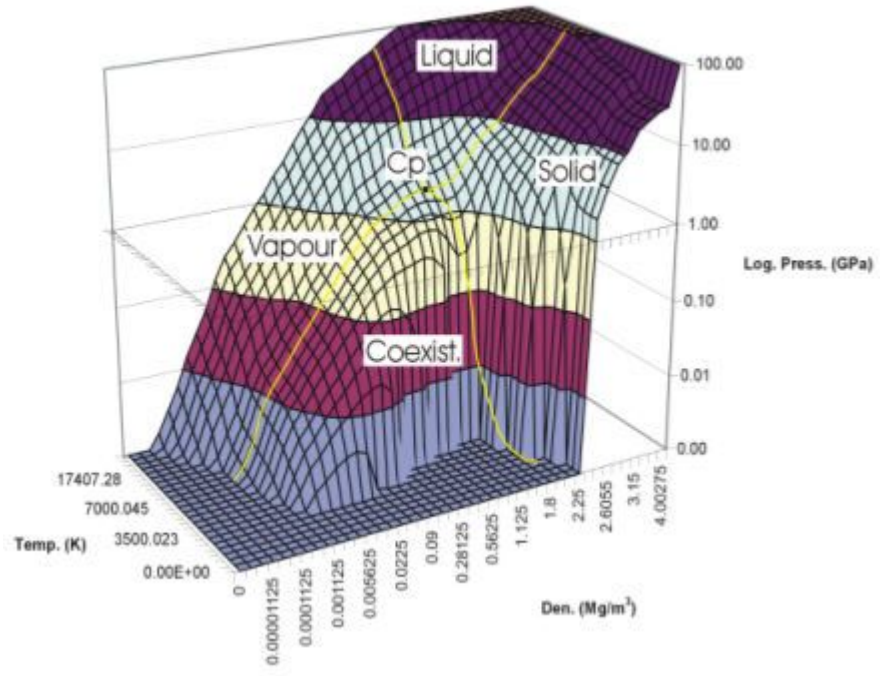
Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

For any other material you will have to supply the required saturation curve data using subroutine EXTAB (the comments in EXTAB explain how to use this subroutine).

Note:

Energy deposition using two phase EOS in Euler may cause instabilities.

22.6.1.4. Sesame EOS (Tables)



The Sesame equation of state tables have been included in Autodyn in the form of a Sesame material library.

This is an extensive library of tables of thermodynamic properties developed and maintained by the Equation of State and Opacity Group of the Theoretical Division of Los Alamos National Laboratory in the USA.

The library currently contains data for over 200 materials including metals, minerals, polymers and mixtures. Most of the tables have data for very wide ranges of density and energy, and are typically used for applications where these wide ranges are required, for example when materials undergo phase changes.

The library format for this EOS is different and you cannot add to or modify the tables, only retrieve data.

In general, our experience suggests that the Sesame tables cannot be used as a black box. The tables contain data for specific ranges in pressure, density, temperature space for each material. Results should be interrogated thoroughly to ensure that simulated material states are consistent with those expected for a particular application.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

If you want to use the Sesame tables in Autodyn, you must obtain the database from LANL (Los Alamos National Laboratory) first. Instructions on how to obtain the database are available here on the Ansys customer site.

The procedure below describes how to use the SESAME database from Los Alamos with Autodyn:

- Obtain the `sesconv.exe` program from Ansys and run this to convert the ASCII SESAME file to a binary format suitable for use with Autodyn.
- Put the resulting `seslib` file in the application folder `C:\Documents and Settings\<usr name>\Application Data\Ansys\251\AUTODYN`.

22.6.1.5. Explosive EOS

The approach required to model the initiation and detonation of explosives and the expansion of the detonation products varies greatly depending on the type of explosive being modeled.

The numerical modeling of the detonation of High Explosives (HE) is well established and precise. In this case, the detonation wave travels at the local sound velocity, ensuring that all the available chemical energy is released before the explosive reacts with its environment. This means that the amount of energy released during the detonation process is fixed and not dependent on the environment (e.g. whether the HE is detonated in air or within a closed container).

This is not the case for other types of explosives (e.g. pyrotechnics, propellants, low density explosives). These explosives burn much slower and consequently, the amount of energy they release depends critically on their environment. This type of burning is referred to as deflagration rather than detonation.

Even with HE, the question sometimes arises as to whether or not the HE is initiated strongly enough for full detonation to ensue.

Autodyn has material models that allow you to simulate all these effects to some degree.

22.6.1.5.1. Lee-Tarver Model

Using the JWL EOS with burn on time provides a good simulation of the detonation process if it is clear that the explosive does indeed fully detonate (release all its chemical energy).

Early experimental work on the initiation of detonation indicated that as a shock front proceeds through a heterogeneous explosive it interacts with local density discontinuities, producing numerous local hot spots that explode but do not propagate.

However energy is released which strengthens the shock so that when it interacts with further non-homogeneities, higher temperatures are produced at these hot spots and more of the explosive is decomposed.

There is thus a run distance for the build-up to detonation and it has been found (Ramsay & Popolato 1965 [6]) that each explosive has a characteristic run to detonation versus shock pressure—the so-called "Pop plots".

Making the further assumption that the explosive passes through the same pressure, distance and state of partial decomposition, regardless of the initial conditions, i.e. that it follows a unique

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

path in distance, time and state space, Forest (Forest 1976 [7]) developed the Forest Fire model to be consistent with the Pop plots by suitable choice of arbitrary coefficients.

This so-called ignition and growth model was not, however, so successful in matching pressure-time data obtained from gauges embedded in the explosive.

A more recent ignition and growth model, the Lee-Tarver model is included in Autodyn.

This model, as with Forest Fire model, is based on the assumption that ignition starts at local hot spots and grows outward from these sites.

The initial version of this model (Lee & Tarver 1980 [8]) described a two-part reaction rate model with a term for ignition of the explosive and a term describing the growth. The reaction rate in this model was given by:

$$\frac{\partial F}{\partial t} = I\left(1-F\right)^{x}\mu^{r} + G\left(1-F\right)^{x}F^{y}p^{z} \tag{22.17}$$

where the compression μ is, as usual,

$$\mu = \frac{\rho}{\rho_0} - 1 \tag{22.18}$$

F was the reaction ratio (ratio of the mass of the gaseous explosive to the total mass of the explosive), p was the pressure in the explosive and I, x, r, G, y, and z were constants. In this model r = 4, so the ignition term depended on the fourth power of the compression of the explosive.

In the more recent model included in Autodyn (Tarver, Hallquist & Erickson 1985 [9]), an additional term is added to overcome the deficiencies of the earlier model in matching data for very short shock pulse initiation.

In this new model, there is a three-fold process of ignition, growth and completion and the reaction rate is given by:

$$\frac{\partial F}{\partial t} = I(1-F)^{b}(\mu-a)^{x} + G_1(1-F)^{c}F^{d}p^{y} + G_2(1-F)^{e}F^{g}p^{z} \tag{22.19}$$

where F, μ and p are defined as above and I, b, a, x, G_1, c, d, y, G_2, e, g and z are constants.

For the explosive PBX-9404, x = 20, so the ignition term depends upon the 20^{th} power of the compression.

The model described by the above equation gives a very rapid pressure spike on ignition, followed by a slow growth of the reaction that accelerates when the regions around the hot spots coalesce.

In Autodyn, the Lee-Tarver equation of state is used to model both the detonation and expansion of high explosives.

The solid inert explosive is modeled using either a JWL EOS or a shock EOS, while the gaseous expansion products are modeled with the JWL EOS with the reaction rate given by the equation above.

For a more physical post-burn behavior for medium and large expansions, the strength of the material is dependent on the burn fraction, F:

$$\sigma = \left(1-F\right)\sigma_0 \tag{22.20}$$

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

where σ is the current yield strength and σ_0 is the original yield strength of the unburned material.

22.6.1.5.2. Powder Burn EOS

In the past, the Powder Burn model has also been known in Autodyn as the Slow Burn material model.

Two different formulations of the powder burn model have been implemented, dependent on the material behavior of the reacted (burned) material:

- A JWL equation of state with constant burn front velocity
- An Exponential equation of state with a burn front velocity that is dependent on the pressure.

The powder burn model is available for the 2D and 3D structural Lagrange, SPH, and Euler multi-material solver types. For Euler, only the formulation using an Exponential reacted equation of state can be used.

There is a need to numerically simulate the combustion of materials where the dominant physical characteristic of the burning is deflagration rather than detonation of the combustible material. The complicated chemical reactions that dictate the burning of the energetic material are extremely difficult to model from first principles. Therefore, a mathematical model based on the averaged quantities of the governing properties has been proposed [10]. The model was developed using detailed studies of a variety of experiments involving energetic materials where the convective burning front moves behind the pressure front of a shockwave. This type of burning occurs in many incendiary devices and munitions.

Theory

The powder burn model is a multi-phase model where gas and solid can be present in one cell at the same time step. The total mass within each cell is found by added together the mass of the gas and of the solid. The volume taken by the gas and solid are both known and therefore the density, compression, etc., of the material within the cell can be calculated. The burning of the material is dependent on the burn fraction *F(t)* where,

$$F(t)=\frac{M_s(t_0)-M_s(t)}{M_s(t_0)} \tag{22.21}$$

and

$$F'(t)=\frac{M'_s(t)}{M_s(t_0)} \tag{22.22}$$

The burning velocity, *c(t)*, is given by a user supplied function, H, dependent on the pressure of the gas,

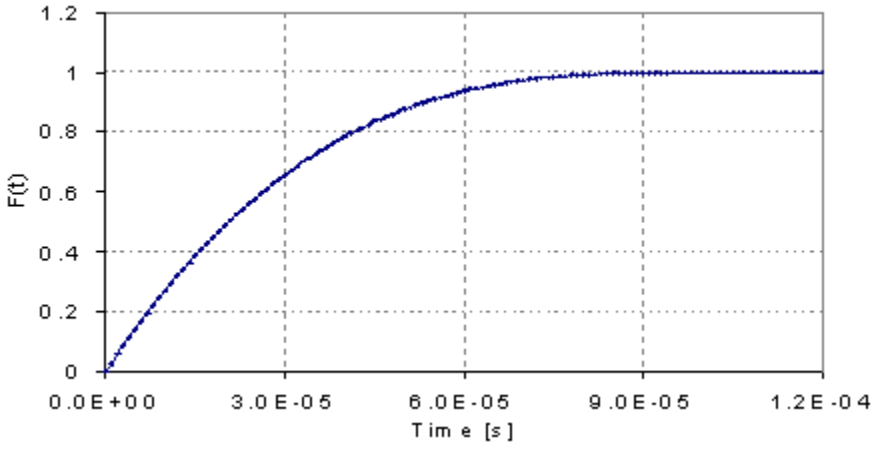
$$c(t)=H(P_g) \tag{22.23}$$

It has been shown [10] that the burn fraction can be represented as a function of H and a function, *G(F(t))*, which expresses the size and shape of the burning surface, such that:

$$F'(t)=G(F(t))H(P_g) \tag{22.24}$$

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

The functions G and H are user supplied and are entered in the material data menu, and a typical burn fraction variation in time is shown below:



The burn fraction should only be calculated on the amount of solid material within each cell which has been ignited. Therefore the total burn fraction within each cell, $F_t(t)$ is found by multiplying the burn fraction, *F(t)*, multiplied by the ignition fraction, *I(t)*, such that

$$F_t(t)=F(t)\cdot I(t) \tag{22.25}$$

The ignition fraction, *I(t)* is dependent on the velocity of the burn front, *V(t)*, which is also known as ignition velocity and which depends on the equation of state selected by the user for the reacted gaseous material.

When the JWL EOS is selected for the Reacted EOS the ignition velocity is taken constant

$$V(t)=C_1 \tag{22.26}$$

and when the Exponential EOS is selected the ignition velocity depends on pressure and density according to:

$$V=C_1+C_2(H(P_g))(1+\gamma(\rho_s)) \tag{22.27}$$

where C_1, and C_2 are user defined constants, *H* and γ are user defined functions and ρ_s is the solid density averaged over the cell volume.

The total burn fraction then dictates the amount of solid which is burnt and has changed into a gaseous state at each time step.

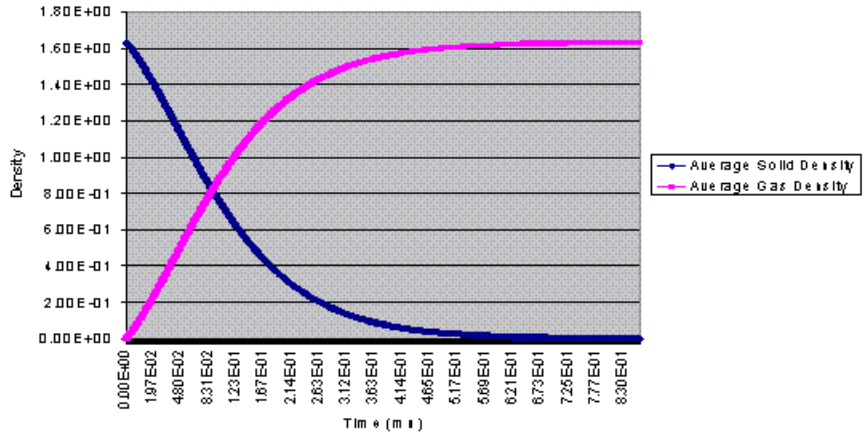
When an Exponential reacted EOS is chosen the pressure of the gas, P_g, is calculated as:

$$P_g=\rho_g e_g Exp(\rho_g/D) \tag{22.28}$$

where ρ_g is the gas density, e_g the gas internal energy and D is a user defined constant.

The pressure of the solid is evaluated using a separate equation of state, which is user defined and can be either a linear or a compaction EOS. As the solid material burns it changes state such that when the cell is fully burned only gas is present within the cell. A typical density variation in time is shown below,

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Material Data Input

Autodyn requires the user to input the values of the burn fraction growth parameters, G and c. These growth parameters are used to evaluate the change in the burn fraction where,

$$\Delta F = G(1-F)^{c} H(P_g) \tag{22.29}$$

When the JWL EOS is chosen for the reacted gaseous material the constant burn front velocity must be entered as well as the relationship between H and P_g. Because the burn velocity is constant, the ignition time of the elements needs to be defined using the on-time detonation logic.

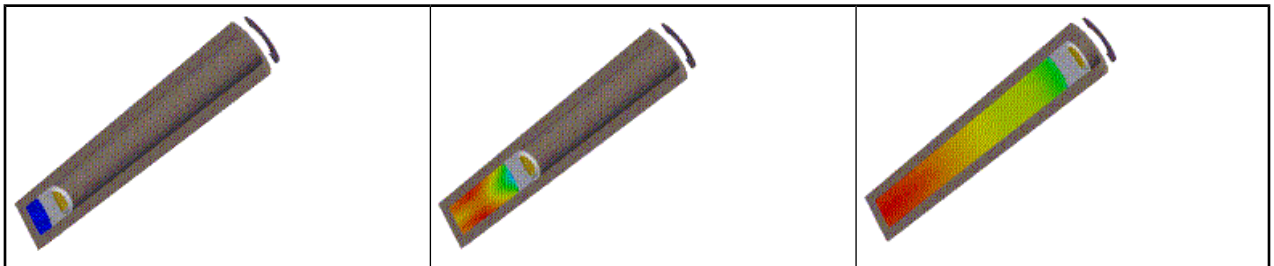
When the Exponential EOS is chosen for the reacted gaseous materials the constants C_1, C_2, and the relationship between H and P_g must be entered in addition to the variation between density and γ in the relationship below:

$$V = C_1 + C_2(H(P_g))(1+\gamma(\rho_s)) \tag{22.30}$$

Because the burn velocity for this reacted EOS is not constant but varies depending on gas pressure and density, the ignition times for the elements cannot be defined using the on-time detonation logic, and to start the ignition of elements the option of manually setting the detonation (ignition) times needs to be used.

The unreacted solid can be represented using a compaction or linear EOS.

Example



Example application of Powder Burn model used in the Multi-material solver to propel a sabot and projectile inside an experimental gun chamber

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

22.6.1.6. Rigid

A rigid material is available under the Equation of State option. Regions of the model filled with such a material will act as a single rigid body, irrespective of whether the individual elements filled are connected to each other. Multiple rigid bodies can be represented by filling different regions of the model with different rigid body materials.

A rigid body defines an un-deformable object that has mass, inertia and momentum. If the mass and inertia parameters are set to zero in the material definition, these quantities will be derived automatically from the material density and the volume of the elements filled. Alternatively, the user can directly specify the body's mass and inertia using the first 10 input parameters.

By default, a kinematic rigid body is defined in Autodyn and its motion will depend on the resultant forces and moments applied to it through interaction with other Parts of the model. Elements filled with rigid materials can interact with other regions via contact or Euler coupling.

A constraint or imposed motion can be applied to a rigid body by selecting the "Constrain rigid body option" in the material definition. Alternatively, a boundary condition can be applied to all the nodes belonging to the rigid body.

Note:

Parts containing any of the new unstructured solvers can be filled with rigid materials. Rigid materials cannot be used in the structured solvers.

22.6.2. Material Strength

While there are many problems which can be calculated using only a hydrodynamic equation of state, there are many applications where material strength effects (i.e. its resistance to shearing forces) cannot be ignored and indeed may even dominate.

If the material is a solid and has finite shear strength then in addition to the calculation of the hydrostatic pressure it is necessary to define relations between shear stress and strain.

We also require relations to define the transition between elastic and plastic strain, both in compression and release, and a relation to define the onset of fracture.

The yield criterion governing the transition from elastic to plastic behavior may involve only a constant yield strength, or this strength may itself be a function of the degree of strain (work hardening), the rate of strain and/or the temperature of the material (energy dependency).

Even in hypervelocity impact problems, where traditionally strength effects could be ignored because of the extremely high stress levels produced, there exist two regimes - far field and late times - where strength effects may be important.

As the impact-created stress levels decay away from the area of impact there may be regions in the target where the local levels become of the order or less than the material strength. If this should be the case the diverging wave would separate out into a two-wave structure, an elastic wave followed by a plastic deforming wave, with consequent differences in material response.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

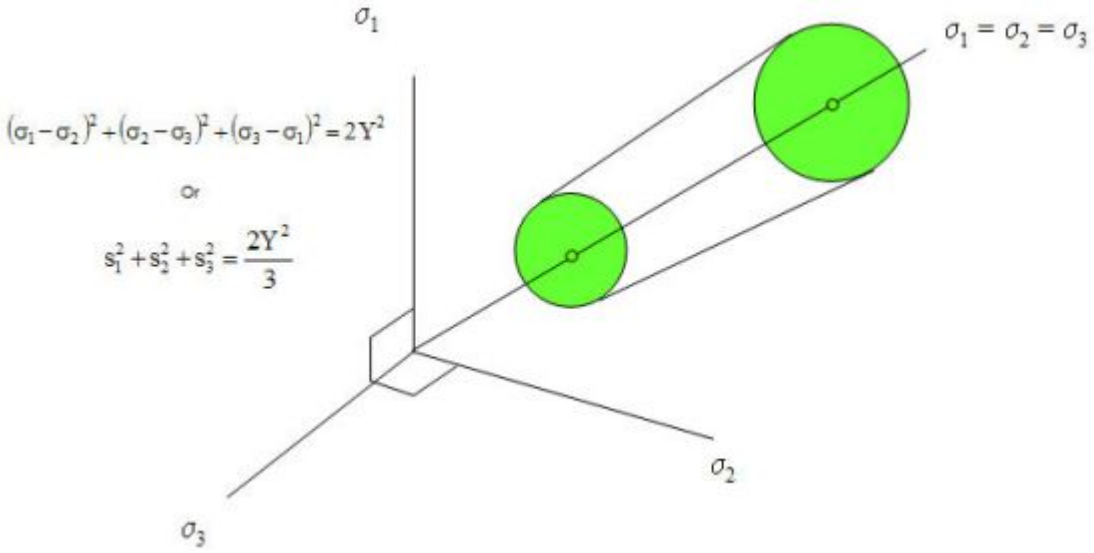
As the projectile is slowed within the target the impact-produced stresses will also decrease and if the target is sufficiently thick a later point in time may be reached when material strength effects cannot be ignored.

It is clear that, unless you can be certain from the outset that material strength effects are not important (because the materials of interest are gases, fluids with no shear strength, or the stress levels in the system under study will always be so high that the strength of the materials could only produce a minor perturbation and may be discounted), the problem should be calculated as one with materials of finite shear strength in order to obtain a realistic solution.

The methodology followed in Autodyn is that first formulated by Wilkins (Wilkins, 1964 [11]) to extend conventional numerical hydrodynamic codes to include the effects of material strength and resistance to shear distortion.

The technique adopted was to work with the stress deviators which are the differences between the total stress and a uniform hydrostatic pressure.

22.6.2.1. Von Mises Model

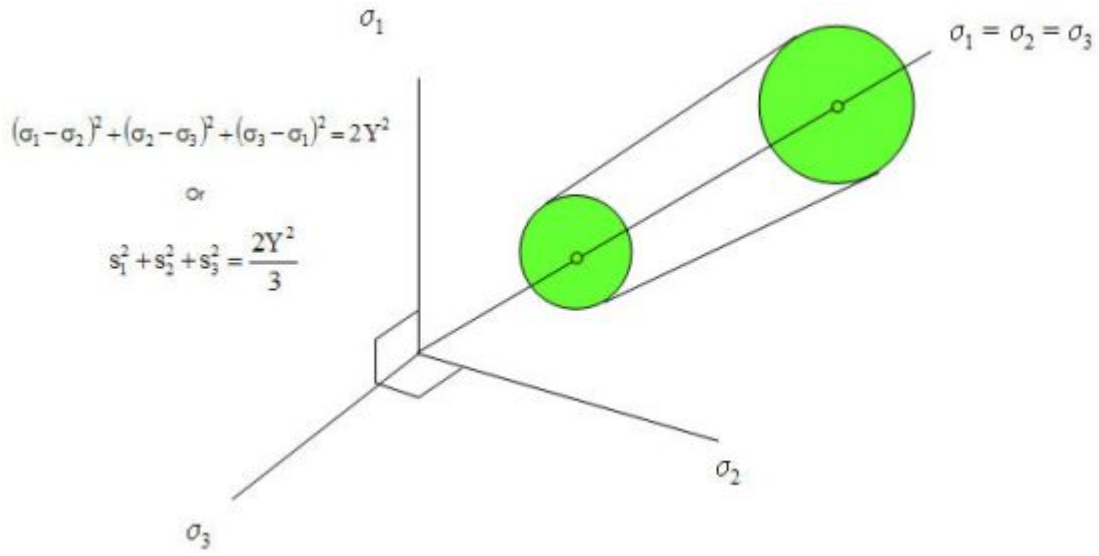


This model uses the original von-Mises premise that the yield stress has a constant value. Consequently, the von-Mises cylinder has a fixed radius. States lying inside the cylinder are elastic. States on the surface of the cylinder are plastic.

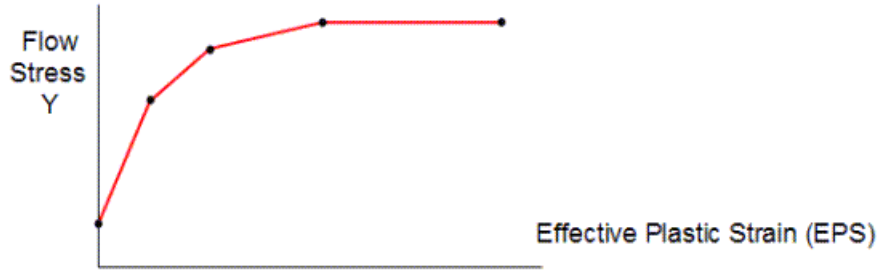
This model does not generally take into account the effect of strain hardening, strain rate sensitivity or thermal softening. However, these effects can be included to some extent by specifying, for example, an average dynamic value for the yield stress. A dynamic enhancement factor of 2 was used very successfully by Wilkins for simulating cylinder tests.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

22.6.2.2. Piecewise JC Model



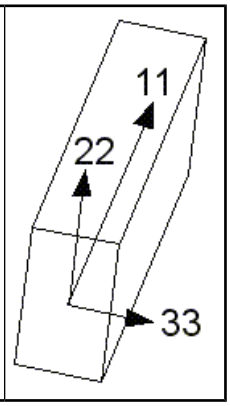
This model is a modification to the Johnson-Cook model, where the dependence on effective plastic strain represented by the term ($A + B\varepsilon_p^n$) is replaced by a piecewise linear function of yield stress Y versus effective plastic strain ε_p.



The strain rate dependence and thermal softening terms remain the same as in the Johnson-Cook model.

22.6.2.3. Beam Resistance Model

The beam resistance strength model lets you define the load-deflection, curvature-moment relationship for beam elements explicitly via input curves. The material data for the beam-resistance model is defined using four 10 point piecewise linear curves.

Axial Force vs. Axial Strain along axis 11 Moment vs. Curvature about axis 11 Moment vs. Curvature about axis 22 Moment vs. Curvature about axis 33	

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

There is no coupling between the different modes of deformation.

The model allows load-deflection data obtained from experiments on structural beams to be fed directly into the software.

The model was developed to allow representation of the global response of reinforced concrete beams/columns subjected to extreme dynamic loading.

22.6.3. Failure Models

The discussions in the sections on strength models explain how plastic yielding arises from the inability of real materials to support arbitrarily large shear stresses. The separation of the flow into separate elastic and plastic flow regimes is described, together with the criteria for recognizing the switch between one regime and the other.

In a similar manner real materials are not able to withstand tensile stresses which exceed the material's local tensile strength.

The computation of the dynamic motion of materials assuming that they always remain continuous, even if the predicted local stresses reach very large negative values, will lead to unphysical solutions.

Some model has to be constructed to recognize when tensile limits are reached, to modify the computation to deal with this and to describe the properties of the material after this formulation has been applied.

Several different modes of failure have been implemented in Autodyn to provide the most suitable form of failure criterion for different types of materials. They can be divided into three categories of failure criteria:

Bulk (isotropic) failure

These models consider the material behavior in an overall (isotropic) manner and allow for failure when some predefined flow variable reaches a critical value.

Directional failure

These models can be used to model failure initiation which is directionally dependent. They are therefore very appropriate for different failure modes such as spalling, plugging, delamination, petalling, and discing. They are only implemented currently for Lagrange, ALE, Shell, and SPH solvers. They cannot be used for Euler since it is not possible to accurately track principal directions in Euler cells.

Cumulative Damage, Johnson-Holmquist Damage

These failure models have been developed to describe the macroscopic inelastic behavior of materials such as ceramics and concrete where the strength of the material can be significantly impaired by crushing. The models can sometimes also be used to follow the behavior of metals which are subjected to tensile stress levels below the ultimate tensile limit but for periods of time long enough to initiate incipient spall.

Post-failure response of materials is isotropic for all of the above failure models.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Following failure, the failed material in that cell can no longer sustain any shear stress or any negative pressures except in the case of the hydrodynamic tensile limit model, where it is assumed that the material "reheals", i.e. while the material pressure is set to zero in the cell for that time step it is assumed that the material can sustain negative pressures down to the hydrodynamic tensile limits in subsequent flow (it may of course reach that limit again and the failure process will be repeated).

In Euler Parts, material(s) in a cell which has failed have their volumes adjusted and void created to satisfy the zero pressure criterion. These materials may be transported into neighboring cells in the following time-step. They will not carry with them any knowledge of the failure in the previous time-step and in the donor cell (other than the values of pressure, volume etc. arising from the failure). The criterion for failure in the recipient cells will test on the updated pressure etc. within those cells without knowledge of past histories.

While some of the different models can be used with all types of solvers, others are implemented in only specific types of solvers. The range of applicability is indicated below:

	Lagrange	**ALE**	**Euler**	**Shell**
Hydrodynamic tensile limit	Yes	Yes	Yes	No
Bulk strain	Yes	Yes	Yes	Yes
Directional failure	Yes	Yes	No	Yes
Cumulative damage	Yes	Yes	Yes	No
Johnson-Holmquist damage	Yes	Yes	Yes	No

22.6.3.1. Material Stress Failure

In this option, and the subsequent two material failure options, the specified directions, referred to as "principal" directions, are defined by the dominant geometric characteristics of the material.

These models are useful for materials which are likely to fail along pre-defined material planes, for example where delamination failure occurs between layers in a composite plate.

Failure will be initiated if any of the principal material stresses or strains exceed their respective specified failure levels.

Because the principal directions are defined by the material directions, there can be a finite, non-zero, shear stress and strain on the planes defined by these directions. Also the maximum direct stresses and strains can lie on planes which are not coincident with the principal material planes, and the maximum shear stress and shear strain may not be on a plane at 45 degrees to the principal material plane.

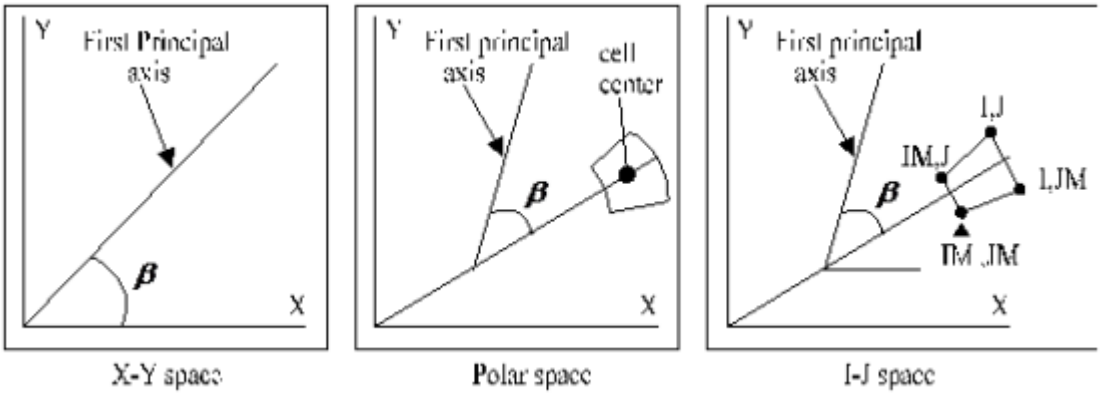
You must specify:

1. Maximum tensile failure stress 11
2. Maximum tensile failure stress 22
3. Maximum tensile failure stress 33
4. Maximum shear stress 12
5. Material axes option

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

6. Rotation angle (degrees)
7. X-origin (Polar space)
8. Y- origin (Polar space)

You define the initial principal material direction through the input parameters (5) to (8) as shown in the following figure:



Failure is initiated if any of the principal material stresses exceed their respective specified tensile failure stress limits. For shear failures the shear stress on planes parallel to the principal directions are checked against the maximum shear stress 12.

22.6.3.2. Material Strain Failure

You must specify:

1. Maximum tensile failure strain 11
2. Maximum tensile failure strain 22
3. Maximum tensile failure strain 33
4. Maximum shear strain 12
5. Material axes option
6. Rotation angle (degrees)
7. X-origin (Polar space)
8. Y- origin (Polar space)

You define the initial principal material direction through the input parameters (5) to (8) as detailed in the previous figure.

Failure is initiated if any of the principal material strains exceeds the respective specified limit. For shear failures the shear strain on planes parallel to the principal directions are checked against the maximum shear strain 12.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

22.6.3.3. Material Stress/Strain Failure

You must specify:

1. Maximum tensile failure stress 11
2. Maximum tensile failure stress 22
3. Maximum tensile failure stress 33
4. Maximum shear stress 12
5. Maximum tensile failure strain 11
6. Maximum tensile failure strain 22
7. Maximum tensile failure strain 33
8. Maximum shear strain 12
9. Material axes option
10. Rotation angle (degrees)
11. X-origin (Polar space)
12. Y- origin (Polar space)

You define the initial principal material direction through the input parameters (9) to (12) as detailed in the previous figure.

Failure is initiated if any of the principal material stresses or strains exceed their respective failure levels. For shear failures the values of the shear stress and strain on planes parallel to the principal directions are checked against the inputted maximum shear stress and strain.

22.6.3.4. Cumulative Damage Model

This model can be used to describe the macroscopic inelastic behavior of material such as ceramics and concrete where the strength of the material can be significantly degraded by crushing.

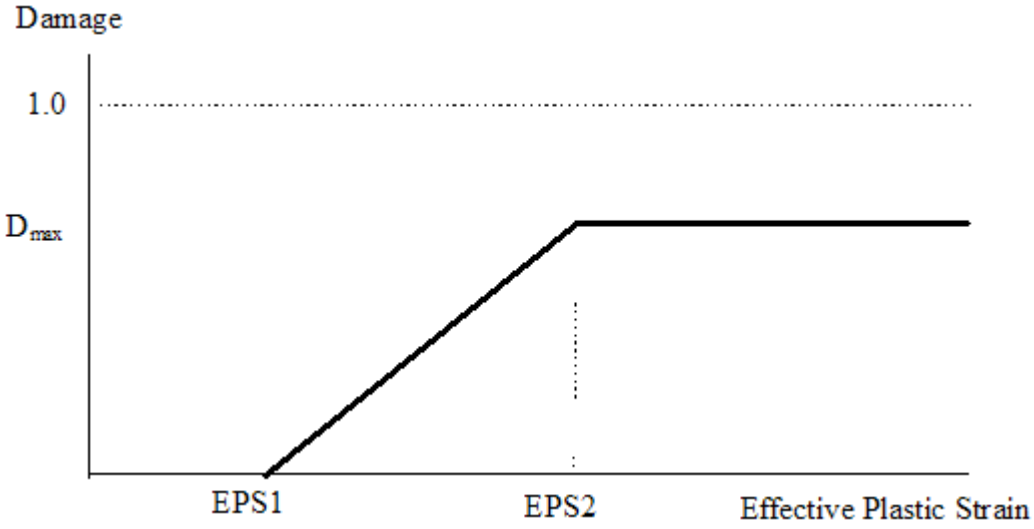
The model can be used only with the Linear equation of state but can be used in conjunction with any strength model (except Johnson-Holmquist which has its own associated cumulative damage model).

However, since experiments indicate that ceramics show a marked increase in compressive strength as the hydrostatic pressure is increased, it is most likely that this model will be used in conjunction with the Mohr-Coulomb model which uses a yield strength that is a function of the local hydrostatic pressure.

To model the progressive crushing and subsequent weakening of ceramic materials the model computes an **r;damage** factor which is usually related to the amount of straining the material is subjected to.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

This damage factor is used to reduce the elastic moduli and yield strength of the material as the calculation proceeds. In the standard model damage is represented by a parameter D which is zero for all plastic deformation for which the effective plastic strain is less than a value EPS1. When the strain reaches a value of EPS1 the damage parameter D increases linearly with strain up to a maximum value D_{max} (<1) at a value of the effective plastic strain EPS2, as shown in the following figure.



Thus

$$D=D_{\max}\left(\frac{EPS-EPS1}{EPS2-EPS1}\right) \tag{22.31}$$

If a different damage function is required you can program this by means of the user subroutine EXDAM.

To describe the progressive crushing of a material the damage function is used to reduce the material's strength. Fully damaged material has some residual strength in compression but none in tension. The current value of the damage factor D is used to modify the bulk modulus, shear modulus and yield strength of the material.

The yield strength is reduced as follows:

- If the hydrostatic pressure is positive

$$Y_{dam}=Y\left(1-D\right) \tag{22.32}$$

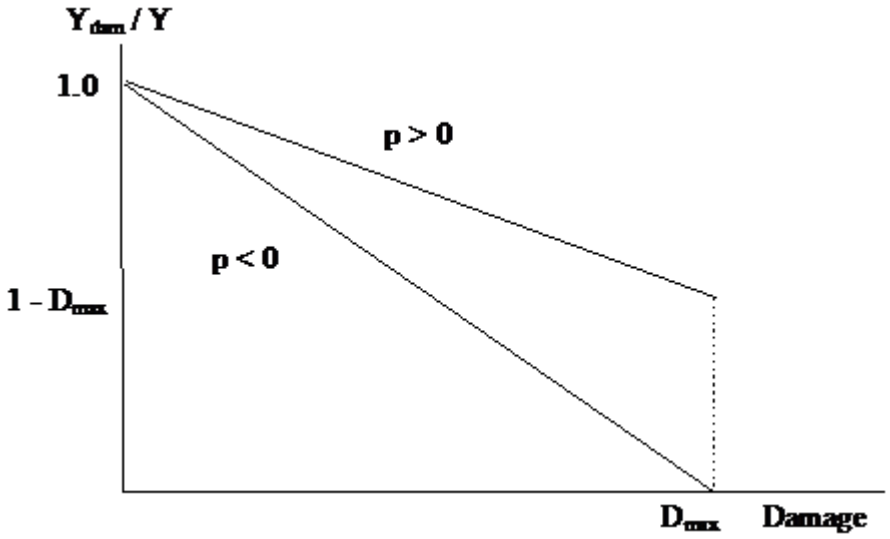
  (providing some residual strength when D reaches its maximum D_{max})

- If the hydrostatic pressure is negative

$$Y_{dam}=Y\left(1-\frac{D}{D_{\max}}\right) \tag{22.33}$$

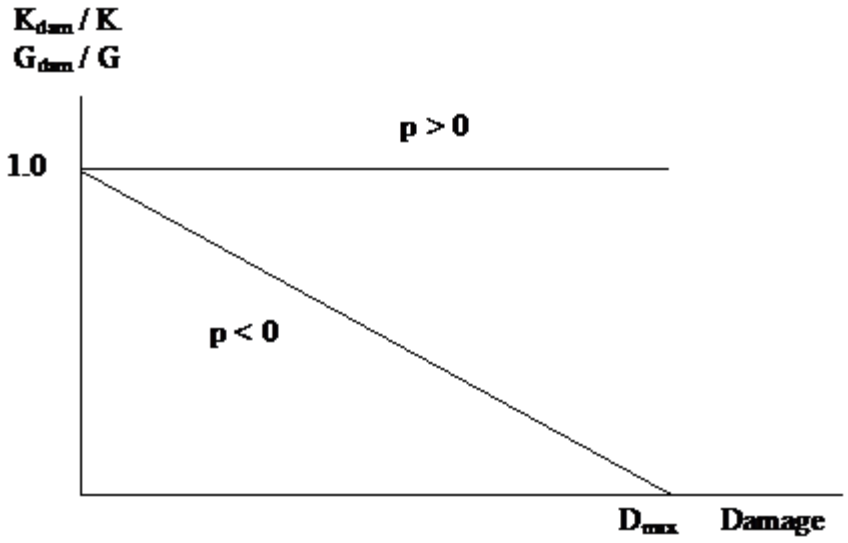
These are illustrated graphically below.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Yield Stress as a Function of Cumulative Damage

The bulk modulus and shear modulus are unaffected in compression, while in tension they are progressively reduced to zero when damage is complete. In tension therefore they are both reduced by the factor $(1 - D/D_{max})$ as shown graphically in the figure below.



Bulk and Shear Modulus as Functions of Cumulative Damage

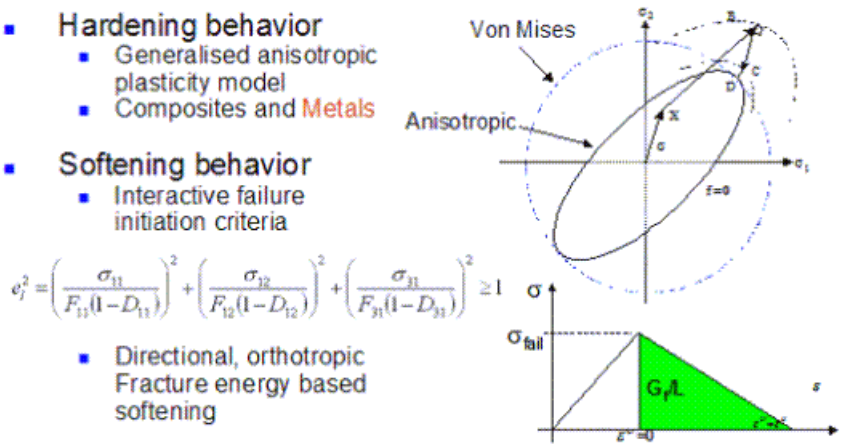
For a more detailed account of the use of this type of model see Persson (1990 [13]).

22.6.3.5. Orthotropic Softening

Advanced composite strength and damage models are available in Autodyn. The Orthotropic Softening Model forms part of this capability and is usually used in combination with an orthotropic equation of state and an orthotropic strength model.

The basic characteristics of these models are described in the figures below.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Outline of new orthotropic yield and damage models for composite materials

Full details of the model can be found in Orthotropic Yield Strength and Hardening Model in the *Autodyn Composite Modeling Guide*, which provides further details of the modeling techniques available and how to obtain/fit material data to the models.

Material data for a Kevlar-epoxy composite for this advanced model is provided in the standard Autodyn material library.

For full details of the original model development and characterization work, please refer to [12].

22.6.3.6. Tsai - Hoffman - Hill Model

The Tsai-Wu, Hoffman and Tsai-Hill criteria can be used with the 3D composite shell element. These models require both compressive and tensile failure strengths, FTij and FCij, to be defined in the plane of the shell which are combined to give a bulk failure criteria as follows;

$$F_{11}\sigma_{11}^2+F_{22}\sigma_{22}^2+2F_{12}\sigma_{11}\sigma_{22}+F_{66}\sigma_{12}^2+F_1\sigma_{11}+F_2\sigma_{22}\geq 1 \tag{22.34}$$

where the constants Fij and Fi are defined for each of the failure models as follows;

Tsai-Wu	Hoffman	Tsai-Hil
$F_{11} = \frac{1}{F_{T11}F_{C11}}$	$F_{11} = \frac{1}{F_{T11}F_{C11}}$	$F_{11} = \frac{1}{F_{T11}^2}$
$F_{22} = \frac{1}{F_{T22}F_{C22}}$	$F_{22} = \frac{1}{F_{T22}F_{C22}}$	$F_{22} = \frac{1}{F_{T22}^2}$
$F_{12} = \frac{-1}{2\sqrt{F_{11}F_{22}}}$	$F_{12} = \frac{-1}{2F_{11}}$	$F_{12} = \frac{-1}{2F_{11}}$
$F_{66} = \frac{1}{F_{T12}^2}$	$F_{66} = \frac{1}{F_{T12}^2}$	$F_{66} = \frac{1}{F_{T12}^2}$
$F_1 = \frac{1}{F_{T11}} - \frac{1}{F_{C11}}$	$F_1 = \frac{1}{F_{T11}} - \frac{1}{F_{C11}}$	$F_1 = 0$
$F_2 = \frac{1}{F_{T22}} - \frac{1}{F_{C22}}$	$F_2 = \frac{1}{F_{T22}} - \frac{1}{F_{C22}}$	$F_2 = 0$

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

22.6.3.7. MO Granular Failure Model

The MO Granular failure model allows a granular material to regain strength once is has failed. Therefore, the failure model can only be used with the MO Granular strength model.

The yield stress is made up of two components, one dependent on the density and one dependent of the pressure,

$$\sigma_Y = \sigma_P + \sigma_\rho \cdot F \tag{22.35}$$

where σ_Y , σ_P, and σ_p denote the total yield stress, the pressure yield stress and the density yield stress respectively. *F* is initially supplied by the user as the Initial Fraction of Yield. Failure occurs if the Von Mises stress is greater than both the total yield stress σ_Y and $F_S P$ where F_S is the user defined variable Slope and *P* is pressure. After material failure, if the current density is greater than the density at which the material failed, *F* is determined by the following equation,

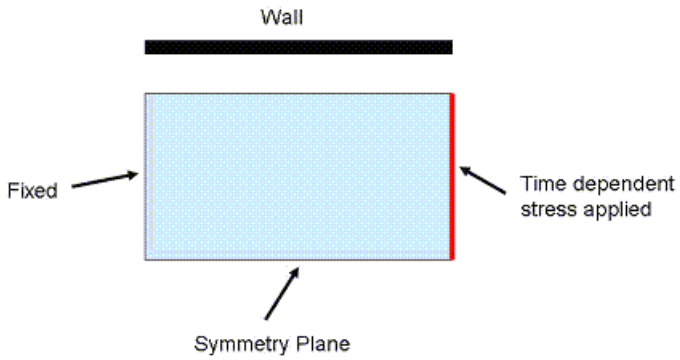
$$F = MAX\left(0, 1 - e^H\right) \tag{22.36}$$

where

$$H = HEXP\left(\frac{\rho_F - \rho}{\rho_F}\right) \tag{22.37}$$

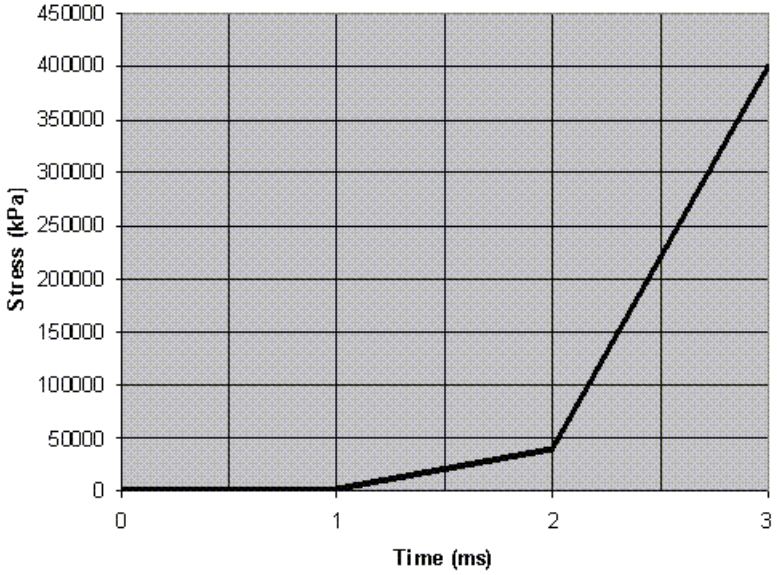
and ρ_F and ρ are the failure density and the current density respectively. *HEXP* is the user defined quantity Heal Exponential and ρ_F is initially supplied by the user as the Initial Failure Density. Therefore *F* is allowed to increase and the material may reheal.

Below is an example of a simulation which uses the MO Granular failure model:

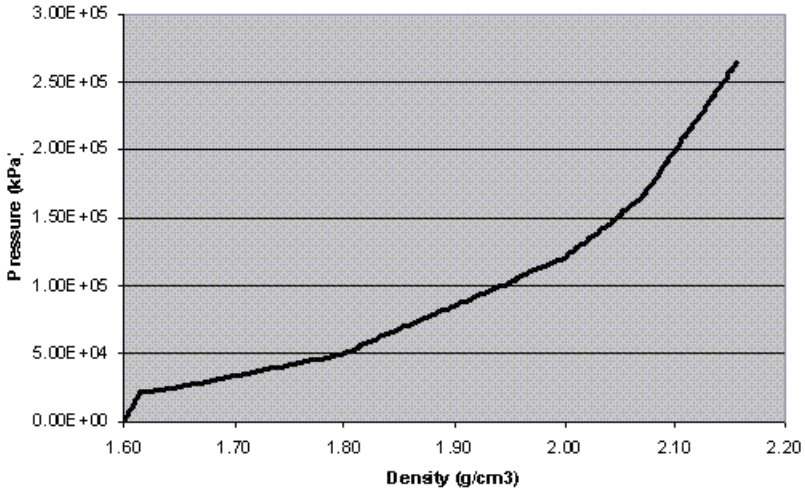


The applied stress is time dependent as shown below:

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information
of ANSYS, Inc. and its subsidiaries and affiliates.

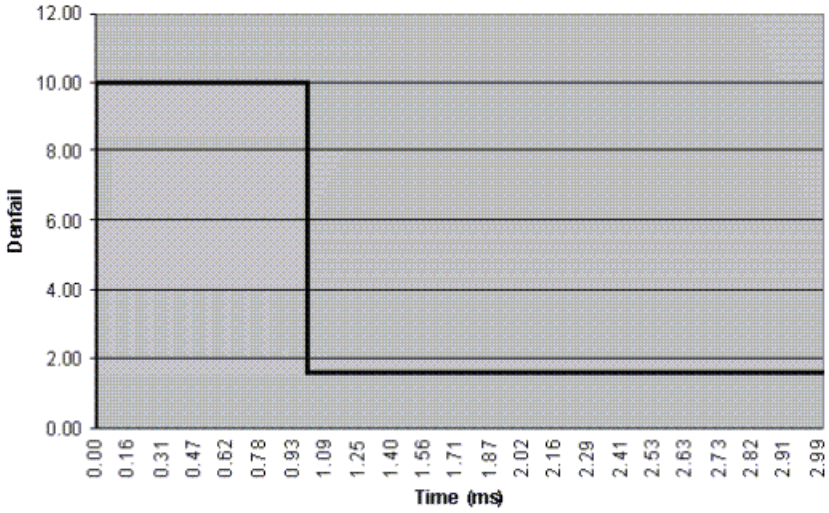


This produces the following pressure density variation:

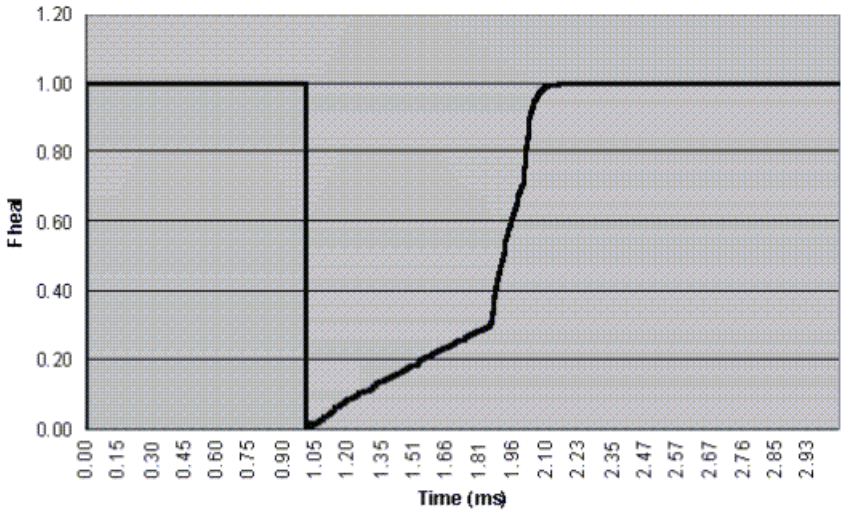


The model undergoes simple compression followed by uni-axial compression forcing the material to fail and then reheal. The initial values used to govern the failure nature of the material are: $F = 1$, $F_S = 3$, $HEXP = 40$, and $\rho_F = 10$. Variations of the governing variables are shown below. The failure density is initially set to 10 and when failure occurs the failure density is set to the current density. Subsequently, as the density increases past this failure density then the material will begin to reheal.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



The fraction of yield, *F*, increases as the density increases until the strength of the material is returned to its initial state:



22.6.4. Material Cutoffs

This option in the Material New/Modify panel lets you change cutoff values that are used by Autodyn. The following fields are available:

Maximum Expansion

This is used to limit the maximum expansion of a cell. If the expansion of a cell exceeds this maximum value, it is reduced to this value.

Minimum Density Factor (Euler)

This cutoff prevents floating point errors due to small quantities of mass in an Euler cell. It is used to ensure that the minimum amount of a particular materials mass in a cell is never less than

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

{density cutoff} * {reference density} * {cell volume}

if {density cutoff} > 0.0

or

- {density cutoff} * {cell volume}

if the {density cutoff} < 0.0

The default value of 1.0E-4 should be adequate for most problems. You might need to change this value if you have a material which expands to well below its reference density (e.g. high explosive expanding into a vacuum or air).

Minimum Density Factor (SPH) and Maximum Density Factor (SPH)

These options let you define minimum and maximum density values for the SPH regions of your problem.

Minimum Soundspeed

This is used to help avoid division by zero in the time step calculation and avoid the possibility of taking the square root of a negative value of C2. If the computed soundspeed is less than this cutoff, it is set equal to the cutoff value.

Maximum Soundspeed (SPH)

This is used to limit the maximum soundspeed for SPH. If the soundspeed exceeds this maximum value, it is reduced to this value.

Maximum Temperature

This is used to limit the maximum temperature within a cell. If the temperature within a cell exceeds this maximum value, it is reduced to this value.

22.7. References

The following references are cited in this appendix:

1. Tillotson, J H (1962). "Metallic Equations of State for Hypervelocity Impact",GA-3216, General Atomic, San Diego, CA, July 1962

2. Allen, R. T. (1967), Equation of State of Rocks and Minerals, General Atomic, San Diego, CA., Report GAMD-7834.

3. Brodie & Hormuth (1966). "The PUFF-66 Computer Program", AFWL-TR-66-48, Air Force Weapons Laboratory, Albuquerque, NM, May 1966

4. Bakken, L H, & Anderson, P D (1969). "An Equation of State Handbook (Conversion Relations between the WONDY/TOODY and the PUFF/KO/HEMP Classes of Shock Wave Propagation Computer Codes)", Sandia Rep. SCI-DR-68-123, Sandia Laboratory, Albuquerque, NM, January 1969

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

5. Morgan, K (1973). "An Expansion Equation of State Subroutine", Comp. Phys. Comm., pp 64-68, 5, North-Holland, 1973

6. Ramsay, J B, & Popolato, A (1965). "Analysis of Shock Wave and Initiation Data for Solid Explosives", pp 233 et seq, Proc 4th Symposium on Detonation, October 1965

7. Forest, C A (1978). "Burning and Detonation", Los Alamos Rep. LA-7245, Los Alamos National Laboratory, Los Alamos, NM, July 1978

8. Lee, E L, & Tarver, C M (1980). "Phenomenological Model of Shock Initiation in Heterogeneous Explosives." Physics of Fluids, 23, 12, pp 2362-2372, 1980

9. Tarver, C M, Hallquist, J O, & Erickson, L M (1985). "Modeling Short Pulse Duration Shock Initiation of Solid Explosives", pp 65 et seq, Proc. 8th Symposium on Detonation, July 1985

10. A Atwood, EK Friis and JF Moxnes, A Mathematical Model for Combustion of Energetic Power Materials, 34th International Conference of ICT, June 24-27, 2003, Karlsruhe Federal Republic of Germany.

11. Wilkins, M L (1964). "Calculation of Elastic-Plastic Flow", pp 211-263 in Methods of Computational Physics, 3, eds. B Alder, S Fernback & M Rotenberg, Academic Press, New York, 1964

12. Riedel W., Harwick W., White D.M., Clegg R.A., "Advanced Material Damage Models for Numerical Simulation Codes, Final Report", EMI-report No. 175/03. ESA Contract No. 12400/97/NL/PA(SC), CCN No. 2.

13. Persson, A (1990). "CM1 - A simple Model for the Dynamic Deformation and Failure Properties of Brittle Materials". Dynamic Research AB., Sweden, 1990

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Index

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information
of ANSYS, Inc. and its subsidiaries and affiliates.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information
of ANSYS, Inc. and its subsidiaries and affiliates.