



© 2025 ANSYS, Inc. or its affiliated companies
Unauthorized use, distribution, or duplication is prohibited.

Release Notes



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 R2
July 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

Global	xvii
1. Advisories	xvii
2. Compatibility with Previous Releases	xviii
3. Installation	xix
4. Licensing	xix
5. Documentation	xx
6. Verification Manuals	xx
6.1. Mechanical APDL Verification Manual	xxi
6.1.1. New Test Cases	xxi
6.1.2. Modified Test Cases	xxi
6.1.3. Removed Test Cases	xxi
6.2. Workbench Verification Manual	xxi
6.2.1. New Verification Test Cases	xxi
6.2.2. Modified Test Cases	xxi
6.2.3. Removed Verification Test Cases	xxi
6.3. Ansys LS-DYNA Verification Manual	xxi
6.3.1. LS-DYNA SOLVE Test Cases	xxi
6.3.2. LS-DYNA EMAG Test Cases	xxi
6.3.3. Modified LS-DYNA Test Cases	xxi
6.3.4. Removed LS-DYNA Test Cases	xxii
7. Online Video Access	xxii
8. Ansys Customer Site	xxii
Ansys 3D Design Products	1
Discovery Application Release Notes	3
1. Discovery Product Advisories	3
2. Discovery Application Enhancements Overview	4
2.1. Fluid Flow Simulation	5
2.1.1. Fans	5
2.1.2. Local Fidelity	5
2.1.3. Fully Developed Flow	6
2.1.4. Backflow Temperature	6
2.1.5. Porous Coordinate System	6
2.2. Structural Simulation	7
2.2.1. Shared Topology Solver Support for Structural	7
2.2.2. Local Coordinate Systems for Structural Post-Processing	7
2.3. Contacts and Interfaces	8
2.3.1. Control of Contact Marker Display	8
2.4. Bolts	8
2.4.1. Custom Bolt Definitions	8
2.4.2. Coaxial Offset Tolerance for Bolt Detection	9
2.5. Transfer to the Mechanical Application	10
2.5.1. Transfer Mesh Settings (Refine)	10
2.5.2. Transfer Heat Transfer Coefficient & Temperature	10
2.6. Solution and Optimization	11
2.6.1. Defined Variations Generator	11
2.6.2. Convergence	11
2.6.3. Reference Pressure	12
2.7. Meshing	13
2.7.1. Mesh Statistics	13

2.7.2. Sharp Edges	14
2.7.3. Improved Structural Meshing Defaults (Refine)	15
2.7.4. Highlighted Failed Tetrahedral Mesh (Refine)	16
2.8. Modeling Tools	16
2.8.1. Improvements to Modeling Documentation	17
2.8.2. Tolerant Volume Extract	17
2.8.3. Improvements to Hybrid Modeling	17
2.9. Text Scale for 3D Annotations	18
2.10. Extend Tool Updated	18
2.11. Wrap Tool Updated	19
2.12. Materials	19
2.12.1. Materials Data for Simulation	19
2.12.2. Materials Data for Simulation - Sample Library	20
2.13. User Interface Enhancements	21
2.13.1. Preview the New UI Theme	21
2.13.2. Grouping Named Selections	21
2.13.3. Editing Physics Conditions	22
2.13.4. Assign Workflow Consistency	22
2.14. User Settings Summary	22
2.14.1. Fluent Solver Settings	22
2.14.2. Reference Pressure Default	23
2.14.3. Import Large File Options	23
2.14.4. Recalculate Rendering	23
2.14.5. Export to the Workbench Application	23
2.14.6. New Export Support: MCNP (Monte Carlo N-Particle)	24
2.15. Results	25
2.15.1. Save Animation to File	25
2.15.2. Heat Flow Monitors in Explore	25
2.15.3. Force Monitors on Fluid-Solid Interfaces	26
2.15.4. Streamlines	26
2.15.5. Max Shear Stress (Tresca)	27
2.15.6. Moment Reaction on Supports	28
2.15.7. Selection of Conditions for Reaction Monitors	28
2.15.8. Reaction Force	28
2.16. Performance	28
2.16.1. Enhanced Graphics Frame Rate and Remote Desktop (RDP) Fix	28
2.16.2. Physics AutoUpdate	28
2.17. In-Product Guidance	29
2.17.1. Search	29
2.17.2. Help Resources	29
2.18. Installation and Licensing	30
2.19. Ansys 3D Design Verification Manual	30
3. Discovery Application Caveats and Known Issues	30
SpaceClaim Release Notes	33
1. SpaceClaim Product Advisories	33
2. SpaceClaim Changes Overview	33
2.1. Installation	33
2.2. Scripting	34
2.3. Import/Export Support	34
DesignModeler Release Notes	35
1. DesignModeler Product Advisories	35

2. DesignModeler Enhancements	35
3. DesignModeler Caveats and Known Issues	35
Ansys Structural Products	37
Mechanical Application Release Notes	39
1. Ansys Engineering Copilot	39
2. Changes in Product Behavior	41
3. Performance	42
4. Add-ons	43
5. General	44
6. Graphics	45
7. Geometry	45
8. External Model	45
9. Contact and Connections	46
10. Mesh	47
11. Additive Manufacturing Process Simulation	47
12. Analysis	47
13. Fracture	49
14. Substructure Analysis	49
15. Structural Optimization	49
16. Motion	50
17. Loads/Supports/Conditions	51
18. Imported Loading Conditions	51
19. Solution	52
20. Explicit Dynamics	53
21. Results	54
22. Scripting and Customization	55
23. Ansys GRANTA Materials Data for Simulation	55
Mechanical APDL	57
1. Structural	57
1.1. Contact	57
1.1.1. Arctangent Friction Model	58
1.1.2. Enhanced Double-Sided Target Surfaces	58
1.1.3. Enhancements to Multipoint Constraints and Assemblies	58
1.1.4. Performance Improvements for MPC Bonded Contacts	58
1.1.5. Enhanced Contact Pair and MPC Surface Based Constraint Generation	58
1.1.6. Generate a Component from Rigid Target Elements	59
1.1.7. Using an Acronym with the RMODIF Command	59
1.2. Elements and Nonlinear Technology	59
1.2.1. Genc Joint Element	59
1.2.2. Joint damping support	60
1.2.3. Direct Element Embedding Enhancement	60
1.2.4. Mono Harmonic Option Support	60
1.2.5. Nonlocal Damage Formulation Support	60
1.2.6. Generalized Remeshing for Adaptive Nonlinear Damage (GRAND)	60
1.2.7. Nonlinear Adaptive Remesh Reasons	60
1.2.8. Adjusting Automatic Time Incrementation	60
1.3. Material and Fracture Modeling	61
1.3.1. Generalized Damage for Brittle Failure Modeling	61
1.3.2. AML Python Module Enhancements	61
1.3.3. Material Curve Fittings Enhancements	61
1.3.4. Small-deflection Hyperelasticity	62

1.3.5. Threshold SIF-range Saved as Result for SMART Fatigue Crack-growth	62
1.3.6. SMART Crack-Growth with Cyclic Symmetry Boundary Conditions	62
1.3.7. Enhancement of J-integral calculation	62
1.4. Linear Dynamics	62
1.4.1. Rotordynamics Analysis	62
1.4.2. Static Multiframe Restarts for Cyclic Symmetry and Multistage Cyclic Symmetry Analyses	63
1.4.3. Node-based Mode File Scoping for Mode-Superposition Analysis	63
1.4.4. Python Script to Calculate Relevant Harmonic Indices for a Multiharmonic Multistage Cyclic Symmetry Analysis.	63
1.4.5. Automatic Inertia Relief	63
1.4.6. Terminating and Restarting a Harmonic Balance Method Analysis	63
1.4.7. Large Mass Method (LMM) Enforced Motion Analysis	63
1.5. Transient Dynamics	64
1.5.1. Adaptive Mass Proportional Damping	64
2. Multiphysics	64
2.1. Acoustics	64
2.1.1. Irregular Perfectly Matched Layer (IPML) for Axially Symmetric Elements	64
2.2. Thermal	64
2.3. Coupled-Field	64
2.3.1. Nonlocal Damage Formulation Support	65
2.3.2. Coupled-field UPFs: Customizable Element Matrices and Load vectors	65
3. Solvers	65
3.1. Distributed-Memory Parallel Processing Enhancements	65
3.2. GPU Acceleration Feature Updates	65
3.3. Updated List of Recommended Devices for the GPU Accelerator	66
3.4. Mixed Equation Solver Enhancements	66
4. Other Enhancements	66
4.1. CAD Support Updates	66
4.2. Element Face Components to Represent Surfaces	67
4.3. New Coded Database File Command	67
4.4. Modified Coded Database File Commands	67
4.5. Element Single-Value Results Supported for SHELL281	67
4.6. Parameter Name Limit Increased to 240 Characters within Mechanical APDL	67
4.7. Component Names Support Special Characters	68
5. Commands	68
5.1. New Commands	68
5.2. Modified Commands	68
5.3. Undocumented Commands	70
6. Elements	70
6.1. New Elements	70
6.2. Modified Elements	71
6.3. Undocumented Elements	71
7. Documentation	72
7.1. Updates for Programmers	72
7.2. Feature Archive	72
8. Technology Showcase: Example Problems	72
9. Mechanical APDL Release 2025 R2 Update Guide	72
9.1. Backward Compatibility	73
9.2. Feature Updates Causing Result or Behavior Changes	73
9.2.1. Change in Harmonic Analysis with CORIOLIS,ON	73

9.2.2. Change in Enforced Motion Calculation	73
9.2.3. Change in Automatic Inertia Relief Behavior	73
9.2.4. Change in MNF File for Special and Service Releases	74
9.2.5. *DMAT: Matrix Type Incompatibility No Longer Accepted for MMF file	74
9.3. Known Incompatibilities	74
9.4. Known Issues	74
9.5. Resolved Issues and Defects	74
9.6. Features Removed or Archived	75
9.6.1. MSAVE	75
9.6.2. KEYOPT 18 for CPT elements	75
Autodyn	77
1. New Features and Enhancements	77
Aqwa	79
1. Aqwa Solver Modules	79
2. Aqwa Graphical Supervisor (AGS)	79
3. Hydrodynamic Analysis Systems	79
Ansys Composite Prep (ACP) Release Notes	81
1. New Features and Enhancements	81
1.1. Transformation Wizard for Rosette Definitions	81
1.2. Symmetric Layups: Copy and Paste in Reverse Order	81
1.3. Automatic Naming of Modeling Plies	81
1.4. 3D Solid Models: Ply Thicknesses at Nodal Locations for CAD Geometries	82
1.5. Lookup Tables: Definition of Variable Data	82
1.6. PyACP Enhancements	82
1.7. Postprocessing through PyDPF Composites	82
1.8. Ansys Composite Cure Simulation (ACCS) Updates	83
2. Known Limitations	83
Material Designer Release Notes	85
1. Notes	85
2. Known Limitations	85
Additive Manufacturing	87
1. LPBF Process Simulations	87
2. DED Process Simulations	88
3. Distortion Compensation	88
4. Changes in Product Behavior	88
5. AM-Related Mechanical APDL Commands	89
Sherlock	91
1. Known Issues	91
2. Scheduled Updates	91
3. General Enhancements and Improvements	91
3.1. Solder Fatigue Model Updates	91
3.2. Performance Improvements	92
3.3. General Enhancements	93
3.4. Other Improvements	93
4. API Updates	94
4.1. New APIs	94
4.2. Modified APIs	94
5. Bug Fixes	94
6. Third Party Dependency Changes	97
Forming	99
1. New Features and Enhancements	99

2. Beta Features	101
Motion Release Notes	103
1. Changes in Product Behavior	103
2. New Features	103
3. Functional Enhancements	103
LS-DYNA Release Notes	105
1. New Documents	105
Ansys Fluids Products	107
Fluent	109
1. Supported Platforms for Ansys Fluent 2025 R2	109
2. New Features in Ansys Fluent 2025 R2	109
2.1. Meshing Mode	109
2.2. Solution Mode	110
2.3. Fluent Workspace Applications	119
3. Updates Affecting Ansys Fluent 2025 R2 Code Behavior	122
3.1. Meshing Mode	123
3.2. Solution Mode	123
3.3. Fluent Workspace Applications	126
4. Features To Be Removed in a Future Release	126
CFX	127
1. Supported Platforms	127
2. New Features and Enhancements	127
3. Updates Affecting Code Behavior	128
CFD-Post	129
1. Supported Platforms	129
2. New Features and Enhancements	129
3. Incompatibilities	129
TurboSystem Release Notes	131
1. Supported Platforms	131
2. New Features and Enhancements	131
TurboGrid	133
1. Supported Platforms	133
2. New Features and Enhancements	133
BladeModeler	135
1. Supported Platforms	135
2. BladeGen	135
2.1. New Features and Enhancements	135
3. BladeEditor	135
3.1. New Features and Enhancements	135
4. BladeBuilder	135
4.1. New Features and Enhancements	135
Polyflow Classic	137
1. New Features	137
2. Supported Platforms	137
3. Resolved Issues and Limitations in Ansys Polyflow Classic 2025 R2	137
Forte	139
1. New Features and Enhancements	139
2. Resolved Issues and Limitations	139
Chemkin	141
1. New Features and Enhancements	141
2. Resolved Issues and Limitations	142

FENSAP-ICE	143
1. New Features and Enhancements in Ansys FENSAP-ICE	143
2. Resolved Issues and Limitations	145
EnSight	147
1. New Features and Enhancements in Ansys EnSight	147
2. PyEnSight	148
3. Ansys Dynamic Reporting (Nexus)	148
4. Beta in This Release	148
5. Alpha in This Release	149
6. Resolved Issues and Limitations	149
FreeFlow Release Notes	151
1. New Features	151
1.1. General New Features (SPH)	151
1.2. General New Features (Solver SDK)	152
1.3. Ansys FreeFlow Coupling with Ansys Mechanical Software	152
2. Improvements	152
2.1. General Improvements	152
2.2. Performance Improvements	152
2.3. Postprocessing Improvements	152
3. FreeFlow Modules	153
4. Beta in This Release	153
5. Resolved Issues and Limitations	154
Ansys Rocky Release Notes	155
1. New Features	155
1.1. General New Features (DEM)	155
1.2. General New Features (SPH)	155
1.3. General New Features (Rocky SDK)	155
1.4. Ansys Rocky Coupling with Ansys Fluent Software	156
1.5. Ansys Rocky Coupling with Ansys Mechanical Software	156
1.6. Ansys Rocky Coupling with Ansys Motion Software	156
2. Improvements	156
2.1. General Improvements	156
2.2. Performance Improvements	157
2.3. User Experience Improvements	157
2.4. Postprocessing Improvements	157
3. Rocky Modules	157
4. Rocky Scripts	158
5. Beta in This Release	159
6. Resolved Issues and Limitations	159
Release Notes: Thermal Desktop 2025 R2 Service Pack 2	161
Ansys Electronics Products	163
Icepak	165
1. Introduction	165
2. New and Modified Features in Ansys Icepak 2025 R2	165
3. Resolved Issues and Limitations in Ansys Icepak 2025 R2	165
Ansys Optical Products	167
Speos Release Notes	169
1. New Features	169
2. Enhancements	170
3. Changes	171
4. Resolved Issues and Limitations	172

Speos for NX Release Notes	173
1. New Features and Enhancements	173
2. Changes	174
3. Resolved Issues and Limitations	175
Speos for Creo Parametric Release Notes	177
1. New Features and Enhancements	177
2. Changes	178
3. Resolved Issues and Limitations	179
Speos Labs Release Notes	181
1. New Features and Enhancements	181
2. Changes	181
3. Resolved Issues and Limitations	181
Speos HPC Release Notes	183
1. New Features and Enhancements	183
2. Resolved Issues and Limitations	183
Ansys Acoustics Products	185
Ansys Sound Release Notes	187
1. Ansys Sound: Analysis and Specification	187
1.1. New Features	187
1.2. Enhancements	187
1.3. Resolved Issues and Limitations	187
2. Ansys Sound: ASDforEV	188
2.1. Enhancements	188
2.2. Resolved Issues and Limitations	188
3. Ansys Sound: Jury Listening Test	188
3.1. Resolved Issues and Limitations	188
Ansys Autonomous Vehicle Products	189
AVxcelerate Lighting and Sensors Release Notes	191
1. Enhancements	191
2. Change	191
AVxcelerate Sensors Release Notes	193
1. New Features	193
1.1. New Rendering Engine for Camera Simulations	193
1.2. Direct Data Memory Access	193
1.3. Radar Phase Noise	194
1.4. Radar Range Filters	194
1.5. Simulation Reset	194
2. Enhancements	195
2.1. Remote Data Memory Access	195
2.2. Thermal Camera Sensor Workflow Enhanced	196
2.3. 3D Bounding Box	197
2.4. Simulation Improvements	198
3. Changes	198
3.1. Supported Operating Systems	198
3.2. Compatibility with driving simulators	198
3.2.1. Co-simulation with SCANer 2023.2	198
3.2.2. CarMaker 11.0 and 11.1 no longer supported	198
3.3. Folder and file names changed	199
Ansys Mesh Prep Products	201
Meshing	203
1. Changes in Product Behavior from Previous Releases	203

2. Enhancements to Mesh Workflows	204
3. Enhancements to MultiZone Method	206
4. Enhancements to Pull	206
5. Enhancements to Weld	206
6. Enhancements to Mesh Details View	207
7. Enhancements to Local Sizing Control Options	207
ICEM CFD	209
Ansys Connect Products	211
Ansys Minerva	213
1. New Features and Enhancements	213
1.1. Installation and Setup	213
1.2. Expression Tester	213
1.3. Data	214
1.4. Applications	214
1.5. Configuration	215
1.6. Jobs	216
1.7. Generic Connector	216
2. Resolved Issues	217
3. Known Issues and Limitations	217
4. Documentation Enhancements	217
Distributed Compute Gateway (DCG)	219
1. New Features and Enhancements	219
2. Resolved Issues	219
3. Known Issues and Limitations	219
optiSLang	221
1. General	221
2. Licensing	223
3. User Interface	223
4. Connectors	230
5. Algorithms	230
Ansys Platform Products	233
ACT	235
Ansys Dynamic Reporting	237
Ansys Viewer	239
1. Advisory: Ansys Viewer End of Life	239
2. New Features and Enhancements	239
3. Resolved Issues and Limitations	239
4. Known Issues and Limitations	239
CAD Release Notes	241
Distributed Compute Services (DCS)	245
DesignXplorer	247
Remote Solve Manager (RSM)	249
1. New Features and Enhancements	249
2. Issues Resolved in this Release	249
3. Known Issues and Limitations	249
System Coupling	251
1. Graphical User Interface	251
2. Participants	251
3. Mapping Accuracy	252
4. Solver Precision	252
5. Expressions	252

6. User-Friendly Mapping	253
7. Terminating Participant Processes	253
8. Reference Frames and Instancing	253
9. System Coupling Infrastructure	253
10. Data Model and Commands	254
11. Participant Library	254
12. Known Issues and Limitations	254
Workbench	255
1. General Workbench Enhancements	255
2. Engineering Data Workspace	255
3. Injection Molding Data	256
Embedded Software Products	257
Scade One Release Notes	259
1. New Features and Enhancements	259
2. Limitations/Known Issues	260
SCADE Suite Release Notes	263
1. New Features and Enhancements	263
2. Certification Credits	263
3. Limitations and Workarounds	264
4. Migration Information	265
5. Tools Version Compatibility	266
SCADE Display Release Notes	267
1. Fixes and Enhancements	267
2. Certification Credits	267
3. Limitations and Workarounds	268
4. Migration Information	269
5. Tools Version Compatibility	270
SCADE Test Release Notes	271
1. New Features and Enhancements	271
2. Certification Credits	271
3. Limitations and Workarounds	272
4. Migration Information	274
5. Tools Version Compatibility	274
SCADE Architect Release Notes	275
1. Fixes and Enhancements	275
2. Limitations and Workarounds	275
3. Migration Information	277
4. Tools Version Compatibility	278
SCADE LifeCycle Release Notes	279
1. New Features and Enhancements	279
2. Certification Credits	279
3. Limitations and Workarounds	280
4. Migration Information	281
5. Tools Version Compatibility	282
SCADE Solutions for ARINC 661 Compliant Systems Release Notes	283
1. New Features and Enhancements	283
2. Certification Credits	283
3. Limitations and Workarounds	284
4. Migration Information	285
SCADE Avionics Package Release Notes	287
1. Fixes and Enhancements	287

2. Limitations and Workarounds	287
3. Migration Information	287

List of Tables

1. Materials Supported by ***MAT_320** 102

Global Release Notes

The release notes are specific to Ansys, Inc. Release 2025 R2 and arranged by application or product, with the exception of:

- [Advisories](#) (p. xvii)
- [Compatibility with Previous Releases](#) (p. xviii)
- [Installation](#) (p. xix)
- [Licensing](#) (p. xix)
- [Documentation](#) (p. xx)
- [Verification Manuals](#) (p. xx)
- [Online Video Access](#) (p. xxii)
- [Ansys Customer Site](#) (p. xxii)

In some cases, installation- and licensing-specific information is detailed within the documentation for a given application or product.

Release notes in printable format (PDF) for this release can be downloaded using the button in the ribbon at the top of the online help page.

Archived Release Notes

Archived release notes for releases prior to 2025 R2 can be found in the online version of [2025 R1 release notes](#) . Or via [ansys.com](#)> Customer Center> Customer Center> General Support> Download Center> Downloads> Product Documentation> Select Release> PDF packages> release.

1. Advisories

- Ansys Viewer 2025 R1 was the final version of Ansys Viewer released by Ansys.

As of the Ansys 2025 R2 release, Ansys Viewer is no longer included in the Ansys Installation program, and support for Ansys Viewer is no longer available.

If you have been using Ansys Viewer for graphics visualization, Ansys encourages you to transition to alternative tools or workflows. If you have any questions or need assistance, contact Ansys Support.

Note that this advisory is *not* referring to the Ansys *Help* viewer, which provides access to Ansys help documentation for customers without internet access.

- Release 2025 R1 will be the final version of the Ansys Additive desktop application, which includes the Additive Print and Additive Science modules. As of this release, no further updates or bug fixes will be provided.

As of the next Ansys release (2025 R2), the Ansys Additive application will be removed from the Ansys Installation program, and support for it will end.

If you use the Ansys Additive application, we encourage you to transition to these alternative tools:

- The Mechanical application, for Additive Print functionality. Several types of geometry-based AM simulations are available, including [laser powder bed fusion](#) (LPBF), [directed energy deposition](#) (DED), [sintering](#), and [distortion compensation](#).
- [PyAdditive](#), for Additive Science functionality. PyAdditive allows you to experiment with process parameters for determining the best combination to use in laser powder bed fusion processes of metal parts. See PyAdditive in the Additive Manufacturing Release Notes.
- Ansys Distributed Compute Services (DCS) 2024 R1 was the final version of DCS released by Ansys. Support for DCS has ended.

The functionality of DCS has been transitioned to Ansys HPC Platform Services — the next-generation solution for HPC workflow management from Ansys. With support for both on-premises and cloud computing environments, HPC Platform Services helps you manage the execution of simulations while making use of your full range of computing assets.

You should now be using HPC Platform Services instead of DCS. For more information, see the **Ansys HPC Platform Services** page on the Ansys Help site or contact Ansys Support.

- Ansys release 2022 R2 (July 2022) was the last release to support native macOS for the Ansys EnSight product.
- The Microsoft Defender Antivirus program in Windows Security, on Windows 10 and 11, will scan all executables associated with the Fluent and Mechanical products and can cause some performance issues. A workaround consists in adding an exception in Windows Defender for the impacted Ansys executables. Note that such procedure shall be operated by authorized IT personnel with full knowledge and understanding of risks.
- In addition to the incompatibilities noted within the release notes, known non-operational behavior, errors and/or limitations at the time of release are documented in the [Ansys Known Issues and Limitations](#). See the [Ansys customer site](#) (p. xxii) or online Help for information about the Ansys service packs and any additional items not included in the **Known Issues and Limitations** document. First-time users of the customer site must register to create a password.
- For a list of issues and limitations in previous releases that have been resolved in Release 2025 R2, refer to the [Resolved Issues and Limitations](#) document on the Ansys Help site.
- Ansys Help (<https://ansyshelp.ansys.com/>) is not supported on the Internet Explorer browser.
- DesignXplorer is deprecated, Ansys recommends using optiSLang containers. The DesignXplorer license (dsdxm license feature) enables comparable functionality for Correlation Analysis, Design of Experiments, Response Surface Modeling, and Optimization.

2. Compatibility with Previous Releases

Backwards Compatibility: Ansys 2025 R2 was tested to read and resume databases from the following previous versions: 2023 R2, 2024 R1, 2024 R2, and 2025 R1. Note that some products are able to read and resume databases from releases prior to 2023 R1. See the specific product sections below for more information. For those products that cannot directly read an earlier database in 2025 R2, first resume it in a supported version and then resume that database in 2025 R2.

Upward/Forward Compatibility: No previous release has the ability to read and resume a database from a more recent release.

3. Installation

The following features are new or changed at Release 2025 R2. Review these items carefully.

- A new `-console` command has been added to the "silent" Ansys Installation. When added to the command line, this argument displays the clickwrap licensing agreement and log files.
- The Ansys installation program has been updated to support Ansys Speos for NX 2506.
- The Ansys Automated Installer includes the option of using your Ansys ID in conjunction with a proxy server.
- The Ansys installation program has been updated to include "reader mode" on supported Linux platforms for select CAD applications.

4. Licensing

The following enhancements were made to Ansys, Inc. Licensing for Release 2025 R2:

General Licensing:

- Ansys Motion Enterprise product now includes access to 4 CPU cores at no additional cost
- The Mechanical Design Type in AEDT enables four Solution Setups:
 - Transient Thermal and Steady-State Thermal requires Icepak Premium or Electronics Enterprise license
 - Modal and Structural requires Mechanical Pro/Premium/Enterprise

Desktop:

- The Ansys License Manager has been updated to FlexNet Publisher client version 11.19.7.0.
- The Ansys License Manager has been updated to OpenSSL version: 3.0.15.
- The Ansys License Manager has been updated to JavaL version: 17.0.14.
- The Ansys License Manager has been updated to Tomcat version: 10.1.39.
- Added support for IPv6.

Web:

- Added access to a range of academic products.
- Added access to Ansys Totem-SC Worker Group.
- Added access to the following Startup Program bundles:
 - Electromagnetics Bundle

- Embedded Software Bundle
 - Optics and Photonics Bundle
 - Semiconductor Analog and Mixed Signal Bundle
 - Semiconductor Digital and AMS Bundle
 - Structural and Fluids Bundle
- The Ansys Licensing Portal has added a new Shared Web Licensing peak service usage report.
 - The Ansys Licensing Portal Web Transactions Table has been enhanced.
 - Paid and unpaid licenses are shown on maximum simultaneous usage charts.

Licensing Portal and ID Portal:

- Column orders are now remembered between sessions.
- Enhancements present consistent actions on Users and Groups pages.

5. Documentation

Ansys Help cannot be launched from Firefox on Linux in a DCV environment for the products CFX, EnSight, and Polyflow.

Ansys Help

Our product documentation is online, directly linked from within our applications. With online documentation, you have access to the best and latest content, updated as soon as it is available. You also gain access to our help, tutorials, and videos in a single, convenient location, accessible from all your Internet-connected devices. Note that, if you are accessing our help directly at <https://ansyshelp.ansys.com/> by logging in with your Ansys Customer Portal user name (email) and password, the site does not currently accept passwords that contain letters topped by umlaut marks.

If you do not have Internet access, or if you would like a local copy of the documentation on your system, you can download an installable version of our product documentation from the [Ansys Download Center](#).

Your feedback is always appreciated as we continue to improve Ansys Help.

Ansys Help (<https://ansyshelp.ansys.com/>) is not supported on the Internet Explorer browser.

6. Verification Manuals

The Verification Manuals for the following products were updated for 2025 R2:

- [6.1. Mechanical APDL Verification Manual](#)
- [6.2. Workbench Verification Manual](#)
- [6.3. Ansys LS-DYNA Verification Manual](#)

6.1. Mechanical APDL Verification Manual

6.1.1. New Test Cases

There are no new Mechanical APDL Verification Manual test cases for 2025 R2.

6.1.2. Modified Test Cases

[VM215](#) was modified to include thermal-electric membrane element [SHELL229](#).

6.1.3. Removed Test Cases

No test cases were removed for 2025 R2.

6.2. Workbench Verification Manual

6.2.1. New Verification Test Cases

No test cases were added for 2025 R2.

6.2.2. Modified Test Cases

No test cases were modified for 2025 R2.

6.2.3. Removed Verification Test Cases

No test cases were removed for 2025 R2.

6.3. Ansys LS-DYNA Verification Manual

Test Case Sets:

[6.3.1. LS-DYNA SOLVE Test Cases](#)

[6.3.2. LS-DYNA EMAG Test Cases](#)

[6.3.3. Modified LS-DYNA Test Cases](#)

[6.3.4. Removed LS-DYNA Test Cases](#)

6.3.1. LS-DYNA SOLVE Test Cases

No new test cases were added for LS-DYNA SOLVE for 2025 R2:

6.3.2. LS-DYNA EMAG Test Cases

No test cases were added for 2025 R2.

6.3.3. Modified LS-DYNA Test Cases

No test cases have been modified for 2025 R2.

6.3.4. Removed LS-DYNA Test Cases

No test cases have been removed for 2025 R2.

7. Online Video Access

To review an extensive library of How-To Videos that detail how to use Ansys product features, go to the [Ansys How-To Videos page](#) at YouTube. Note that you can now also access the How-To videos from the Ansys Help home page for your product.

8. Ansys Customer Site

If you have a password to the [Ansys customer site](#) (www.ansys.com/customercommunity), you can view additional documentation information and late changes. The customer site is also your source for Ansys, Inc. software downloads, service packs, product information (including example applications, current and archived documentation, undocumented commands, input files, and product previews), and online support.

All product documentation is available in printable format (PDF). You can also copy content from within the files into a word-processing program.

Ansys customer site access points:

- **Tutorials and input files**

To access tutorials and their input files, go to the [tutorials area](#) of the customer site.

- **Documentation**

To access documentation files, go to the [documentation area](#) of the customer site.

- **General information**

For general information about materials and services available to our customers, go to the [main page](#) of the customer site.

Ansys 3D Design Products

Release Notes

Release notes are available for the following Ansys 3D Design products:

[Discovery \(p. 3\)](#)

[SpaceClaim \(p. 33\)](#)

[DesignModeler \(p. 35\)](#)

Discovery Application Release Notes

Release 2025 R2 of the Discovery application contains the following.

1. [Discovery Product Advisories](#)
2. [Discovery Application Enhancements Overview](#)
3. [Discovery Application Caveats and Known Issues](#)

1. Discovery Product Advisories

Be aware of the advisories and product changes described here.

Installation and Licensing Advisories

- Operating System Support
 - Support for Microsoft Windows Server 2019 has been dropped.
 - Support for Microsoft Windows Server 2025 has been added.
 - Support for Microsoft Windows 11 version 24H2 has been added.
- Use of the [Defined Variations Generator \(p. 11\)](#) to create defined variations automatically based on an algorithm requires the optislang_level1 or dsdxm licensing increment. All three Discovery application license levels (Pro, Premium, and Enterprise) began including the dsdxm licensing increment in early 2025. If you are using an older license key and want to use this functionality, you may need to request a new key.
- The Discovery application 2025 R2 will be the final version that includes support for NVIDIA Kepler-based and Maxwell-based graphics cards. Starting with Release 2026 R1, the application will only support Pascal-based cards and later versions.
- In general, the Discovery application maintains support of cards for as long as NVIDIA drivers and SDKs allow. For a list of NVIDIA cards no longer supported, see [Installation Prerequisites](#).

SpaceClaim Application Advisories

- API version V23 has been removed at Release 2025 R2.

If you have customized add-ins or scripts, make sure they are upgraded to the most current version.

- The SpaceClaim application has been removed from the Ansys unified installer and the standalone Discovery application installation, but you can still install the SpaceClaim application using any of these methods:

- Download the standalone SpaceClaim application's installation package from the Customer Portal and install it
- Use the command line option **-install_spaceclaim**
- Automatically as a component of a Material Designer application or Speos application installation

Discovery Application Beta Features

For information about Beta features, see *Discovery Application Beta Features Documentation* in the Ansys Help.

Also see [Discovery Application Enhancements Overview \(p. 4\)](#) for features that have been added or enhanced.

2. Discovery Application Enhancements Overview

For product advisories and changes to the Discovery application features that existed in earlier releases, see [Discovery Product Advisories \(p. 3\)](#).

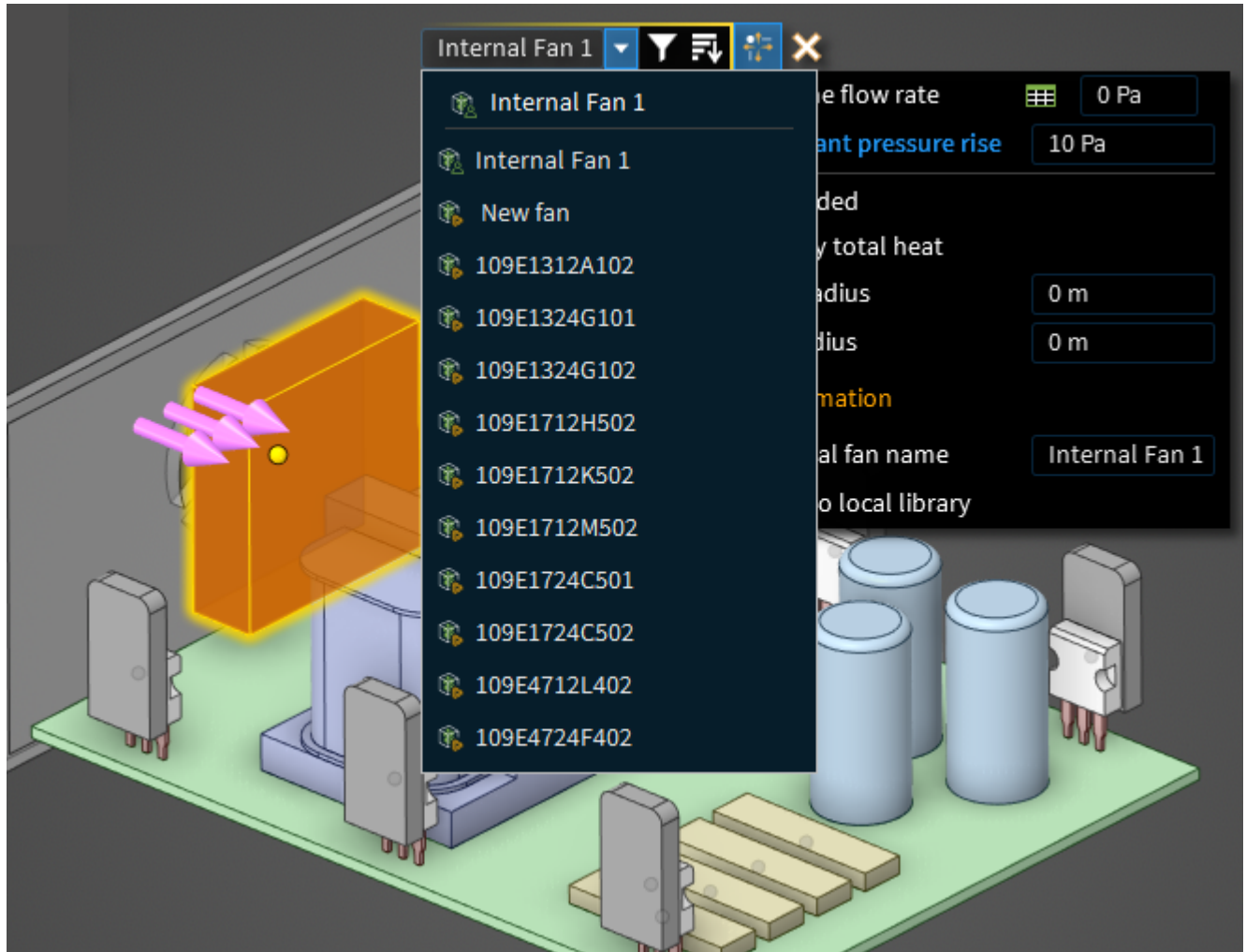
Areas where you will find new capabilities and enhancements in the application include:

- [2.1. Fluid Flow Simulation](#)
- [2.2. Structural Simulation](#)
- [2.3. Contacts and Interfaces](#)
- [2.4. Bolts](#)
- [2.5. Transfer to the Mechanical Application](#)
- [2.6. Solution and Optimization](#)
- [2.7. Meshing](#)
- [2.8. Modeling Tools](#)
- [2.9. Text Scale for 3D Annotations](#)
- [2.10. Extend Tool Updated](#)
- [2.11. Wrap Tool Updated](#)
- [2.12. Materials](#)
- [2.13. User Interface Enhancements](#)
- [2.14. User Settings Summary](#)
- [2.15. Results](#)
- [2.16. Performance](#)
- [2.17. In-Product Guidance](#)
- [2.18. Installation and Licensing](#)
- [2.19. Ansys 3D Design Verification Manual](#)

2.1. Fluid Flow Simulation

2.1.1. Fans

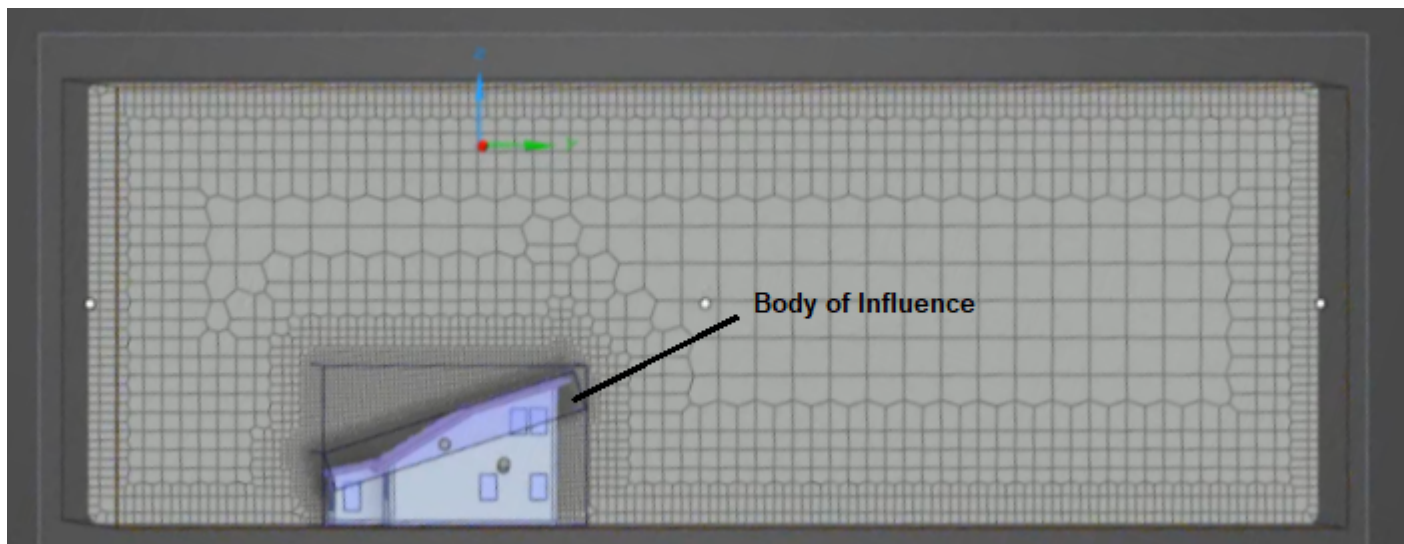
The internal (or 3D) fan condition is available for you to simulate the effect of an axial fan by applying a distributed momentum source to a body within a fluid region. By default, a shrouded fan is created. The internal fan condition supports local and Icepak libraries of fan curve data from several manufacturers. Icepak fan libraries only support gases.



Intake and exhaust (2D) fans and internal (3D) fans can now be solved for both liquids and gases. In previous releases, only gases were supported. However, Icepak fan libraries only support gases.

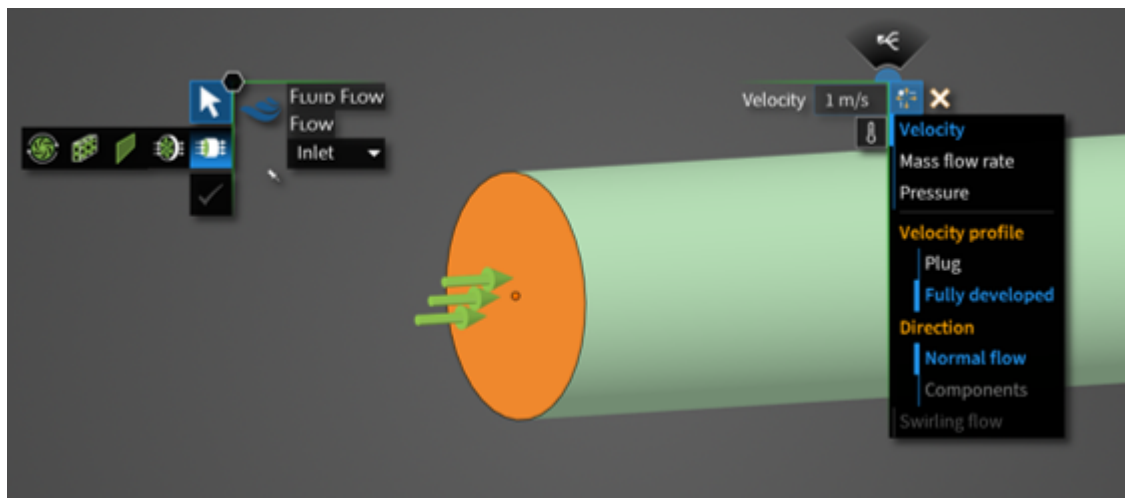
2.1.2. Local Fidelity

Local fidelity can now be applied to a suppressed body that overlaps the fluid domain (body of influence) in **Explore**. Previously, local fidelity could not be used on bodies excluded from the simulation. The fluid region uses this local fidelity in the intersecting region.



2.1.3. Fully Developed Flow

Fully developed velocity profile option for circular velocity inlets is now available in **Explore**. The laminar or turbulent inlet profile is determined from the turbulence model.

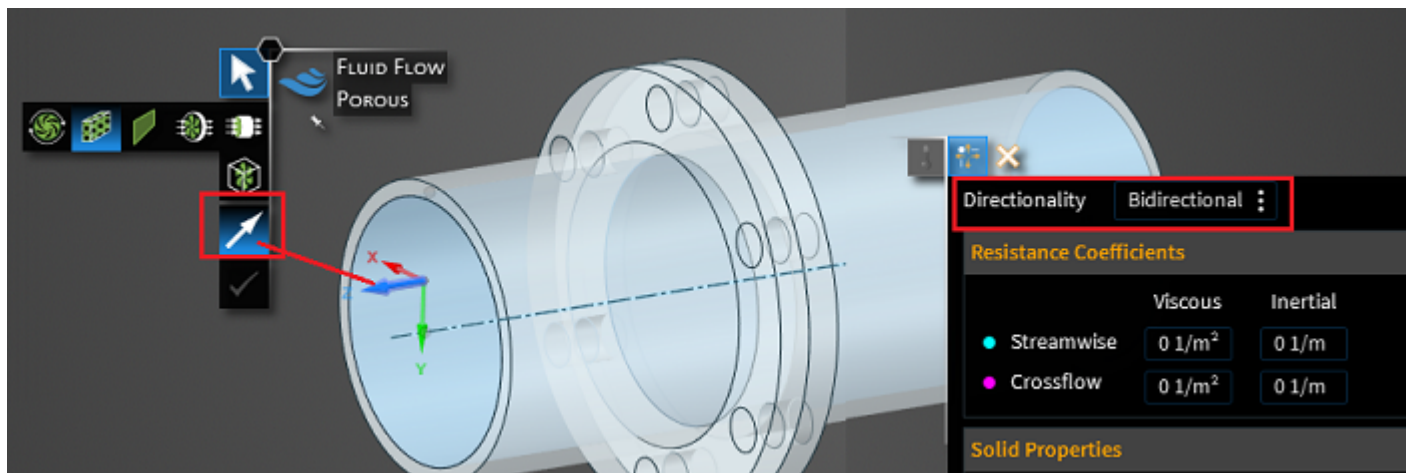


2.1.4. Backflow Temperature

Backflow temperature for fluid flow outlets is now supported in **Explore** and will no longer be ignored in the simulation.

2.1.5. Porous Coordinate System

For bidirectional or orthotropic media, you can use the Direction tool  in the HUD (or Alt-select) to set the direction by selecting a coordinate system or any of the coordinate system's axes.



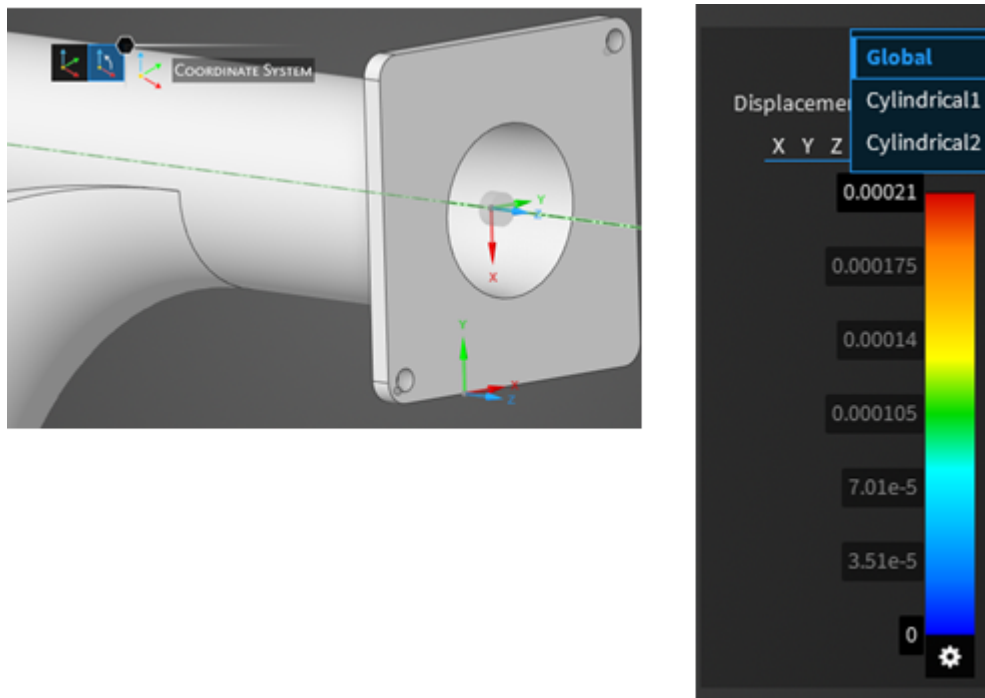
2.2. Structural Simulation

2.2.1. Shared Topology Solver Support for Structural

In structural simulations, simulations with shared topology are now supported by the solver. Any shared topologies are added in the tree under **Connections** → **Contacts**. In the **Explore** stage, shared topology is treated by the solver as bonded contact. In the **Refine** stage, shared topology will create a conformal mesh.

2.2.2. Local Coordinate Systems for Structural Post-Processing

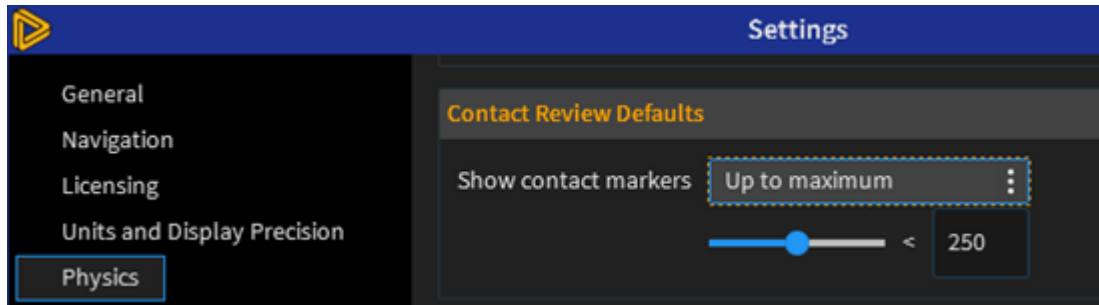
The Discovery application now supports local coordinate systems, both Cartesian and cylindrical, for structural post-processing. You can select a defined local Cartesian or cylindrical coordinate system when viewing contours and isosurfaces and when setting up monitors. This feature enhances post-processing for pressure vessels and structures requiring hoop, radial and axial component results.



2.3. Contacts and Interfaces

2.3.1. Control of Contact Marker Display

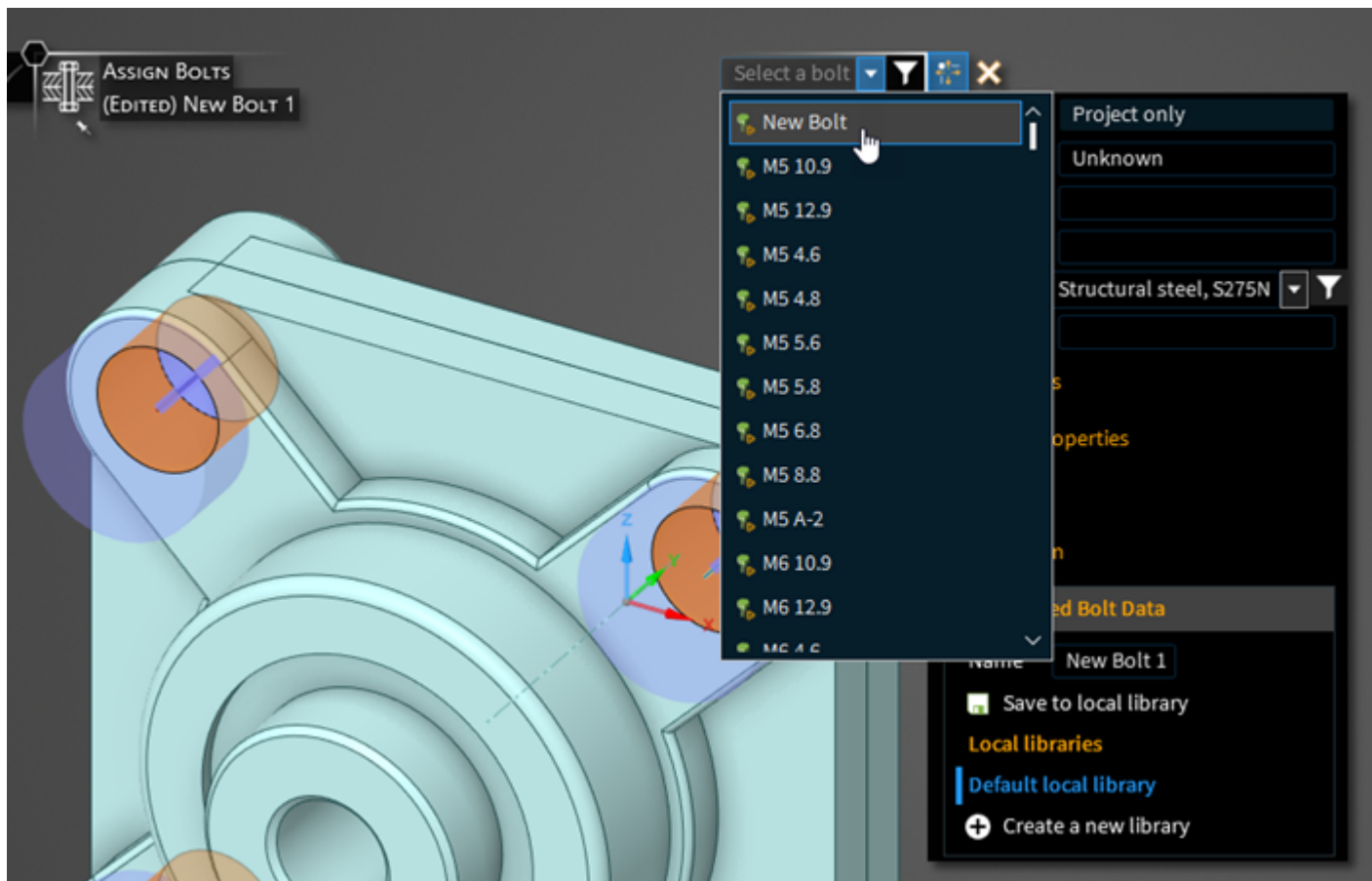
You can now control the display of contact markers in the **Contact Review** tool. You can choose to display all contact markers, markers only on a selected geometry, or markers up to a specified maximum amount. This feature is particularly useful for projects with a large number of contacts.



2.4. Bolts

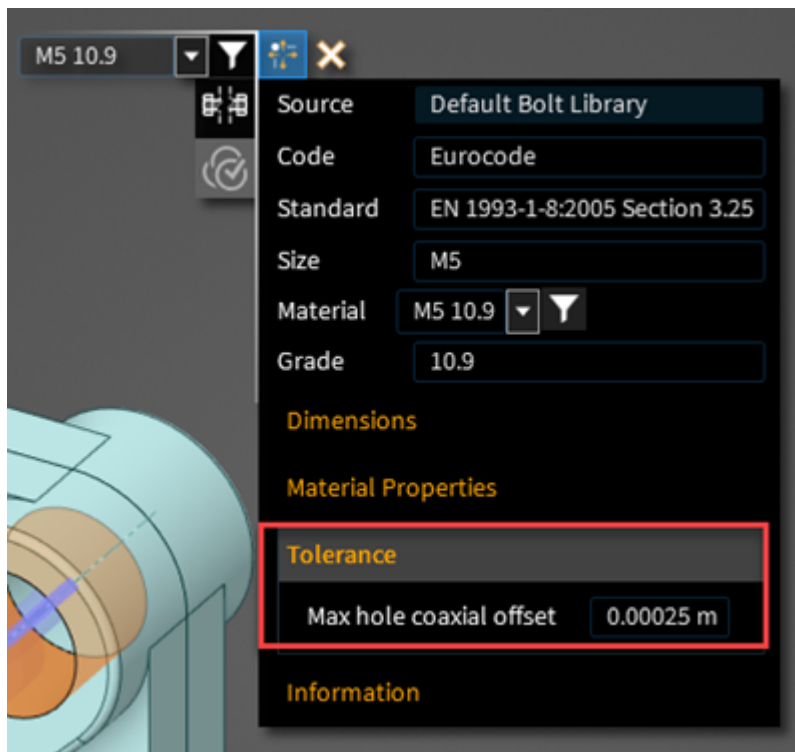
2.4.1. Custom Bolt Definitions

You can now create a new bolt definition, rather than relying on standard library definitions. You can either select and then edit a standard library definition or create a new custom bolt type, then define custom size, material and bolt properties and save custom bolts to local libraries. You can then transfer the bolt data to the Mechanical application from either **Prepare** or **Simulation** tab.



2.4.2. Coaxial Offset Tolerance for Bolt Detection

The Discovery application can now detect bolts with misaligned geometry. To ensure bolts with misaligned geometry are detected, you can specify a maximum tolerance for the bolt hole coaxial offset. The tool can then detect bolts in coaxial holes with small offsets.



2.5. Transfer to the Mechanical Application

2.5.1. Transfer Mesh Settings (Refine)

Mesh settings can now be transferred to the Mechanical application. Both global and local fidelity mesh settings are now available in the Mechanical application.

By default, a Discovery application mesh is transferred as "Read Only" and local fidelity controls are transferred as suppressed. This enables model setup and meshing in the Discovery application with the transfer of mesh and mesh settings to the Mechanical application.

2.5.2. Transfer Heat Transfer Coefficient & Temperature

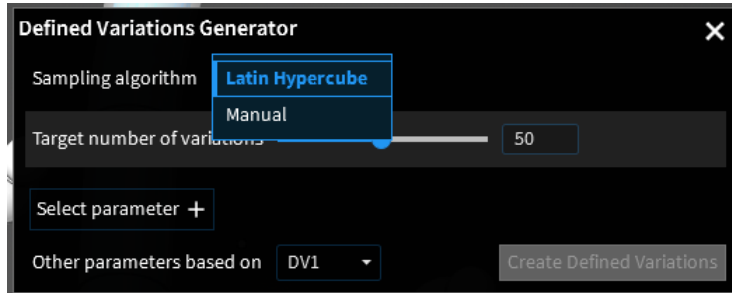
A fluid-solid (conjugate) heat transfer simulation in the Discovery application can now transfer simulation results to either a thermal or thermal-stress simulation in the Mechanical application. It provides realistic and consistent heat transfer coefficient values and near-wall fluid temperatures on fluid-solid interfaces and improves the accuracy of thermal and thermal-stress simulations.

Simulations that include fluid-solid (conjugate) heat transfer will not transfer the fluid or fluid thermal conditions when transferring to the Mechanical application. Instead, the temperature and heat transfer coefficient data at the region interface is transferred to the Mechanical application as an imported load. For the thermal data at the region interface to be included in the transfer, simulation results must be available, even if paused or stalled. For the **Refine** stage, only simulations solved using LiveGX can transfer the thermal data at the region interface.

2.6. Solution and Optimization

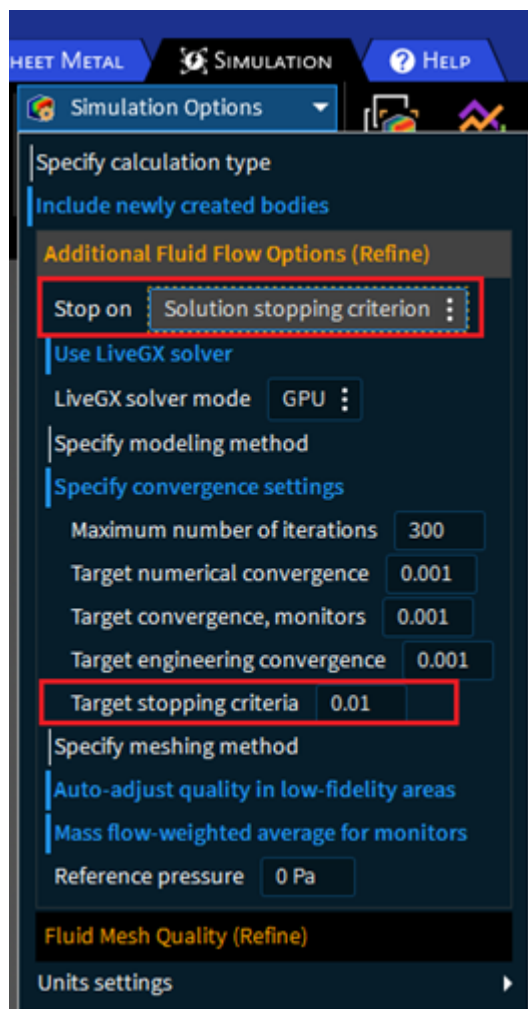
2.6.1. Defined Variations Generator

The **Defined Variations Generator** creates defined variations automatically based on an algorithm or manually based on a range. Using the generator is advantageous in discovering how subtle changes can optimize your simulation.



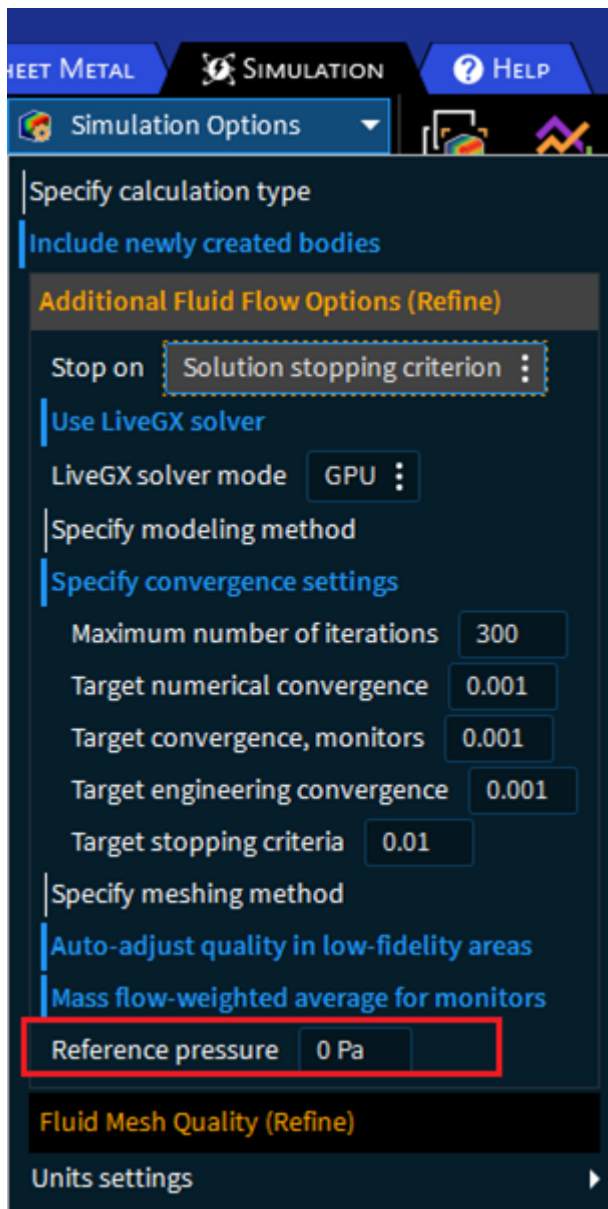
2.6.2. Convergence

You can now specify a **Solution stopping criterion** in **Refine** under **Simulation Options**. A default value of 0.01 (or 1%) is used for the **Target stopping criteria**. However, you can specify your own tolerance. You can also set a default **Target stopping criteria** for the **Solution stopping criterion** in **File Menu > Settings > Physics > Solver Settings** under **Simulation Options Defaults (Refine)** for fluid flow.



2.6.3. Reference Pressure

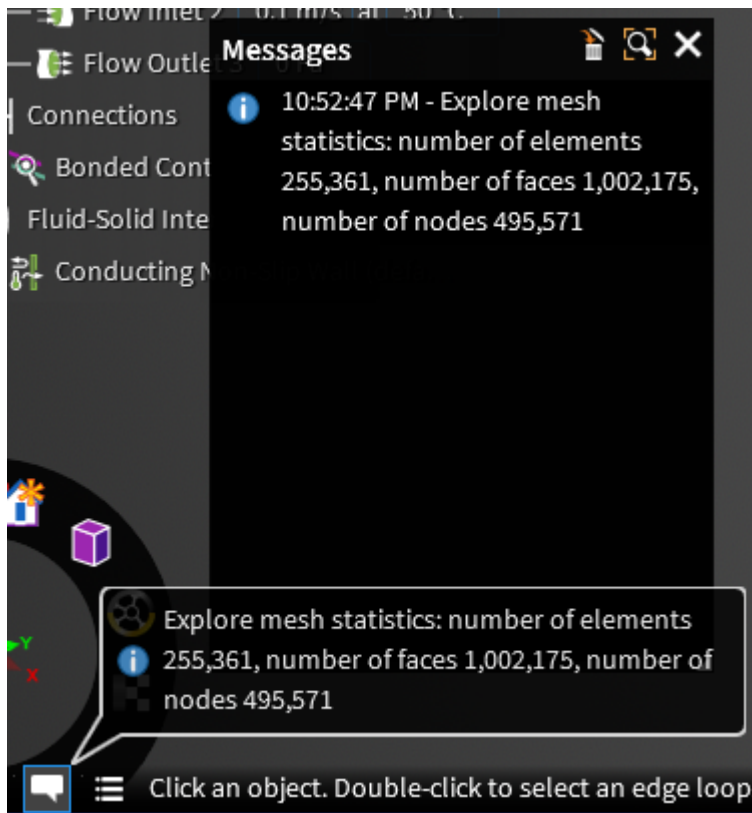
Reference pressure is a new option under **Simulation Options**. The reference pressure you specify is used to adjust the calculation of the force on a wall. You can also set a default Reference pressure in **File Menu > Settings > Physics** under **Simulation Options Defaults (Refine)** for fluid flow.



2.7. Meshing

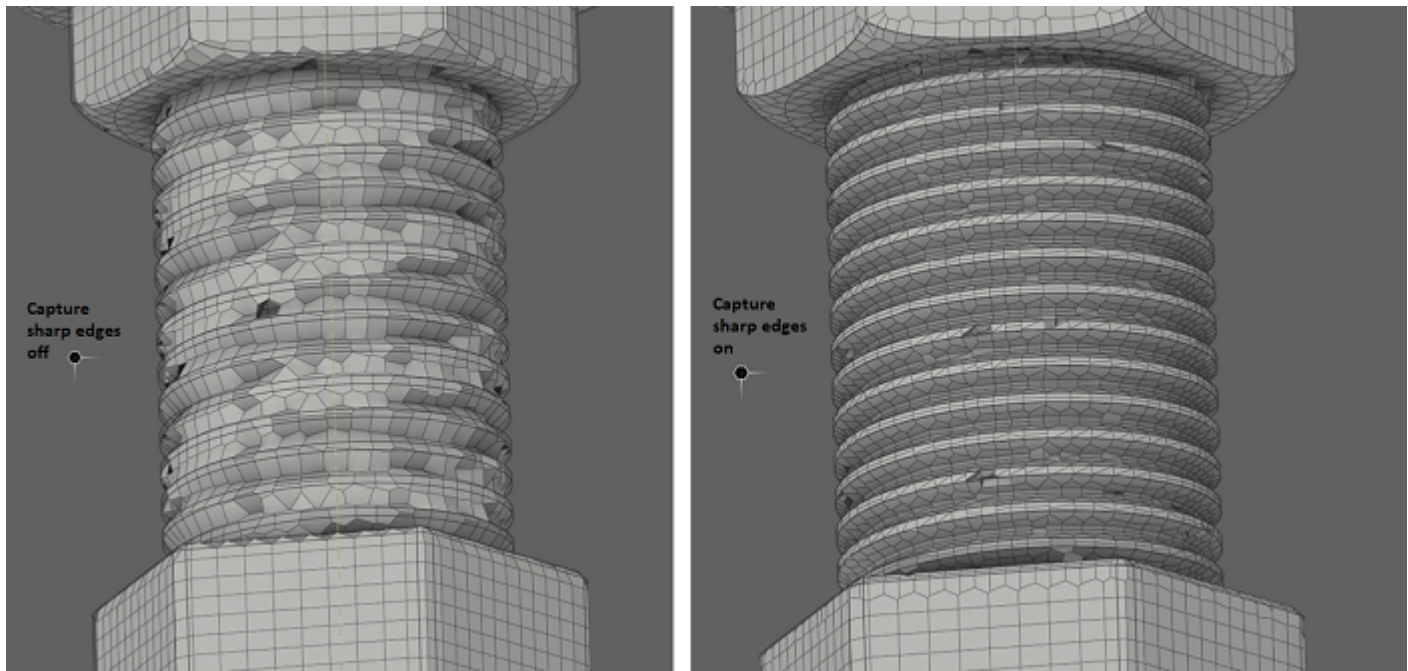
2.7.1. Mesh Statistics

After meshing a fluid flow simulation in **Explore**, you will now see an operational message showing the mesh statistics. This is a similar message to what is available after meshing in **Refine**.



2.7.2. Sharp Edges

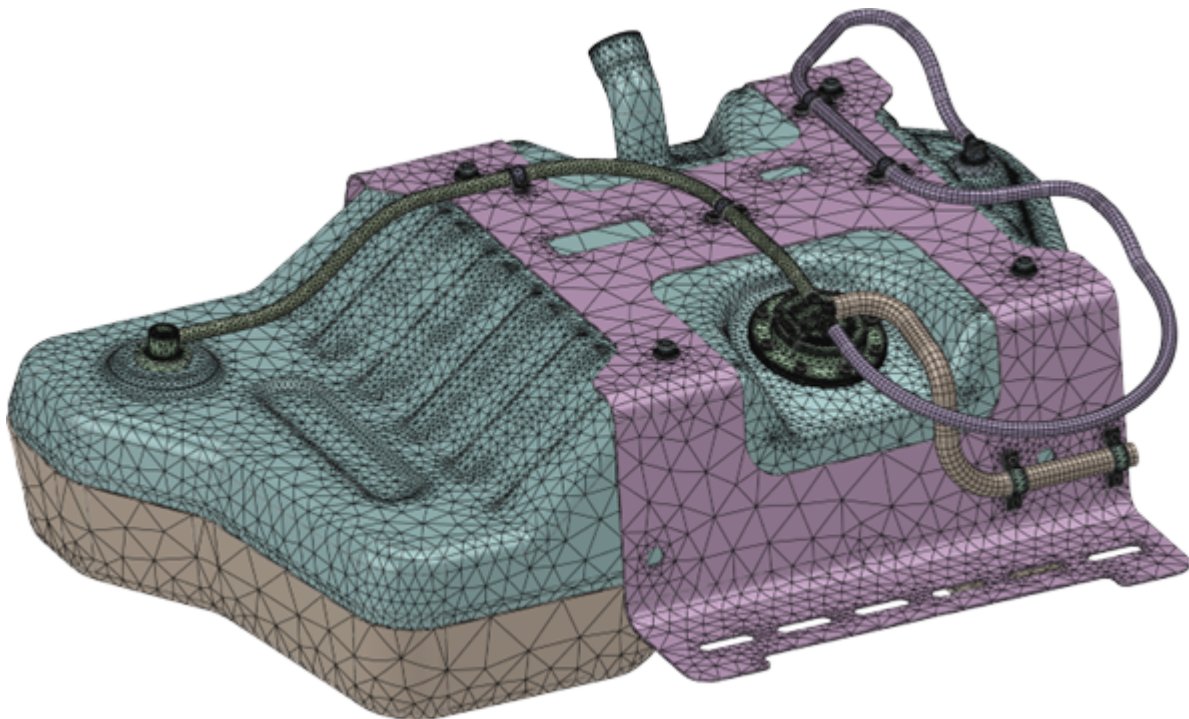
You can now capture sharp edges in your fluid flow simulation model using the **Capture sharp edges** under **Simulation Options** settings in **Explore**. This can improve simulation accuracy especially at inlets, fluid-fluid interfaces, and fluid-solid interfaces. All sharp edges will attempt to be resolved. Without this setting, sharp edges will be defeatured during meshing. Resolving the sharp features can result in additional computational expense.



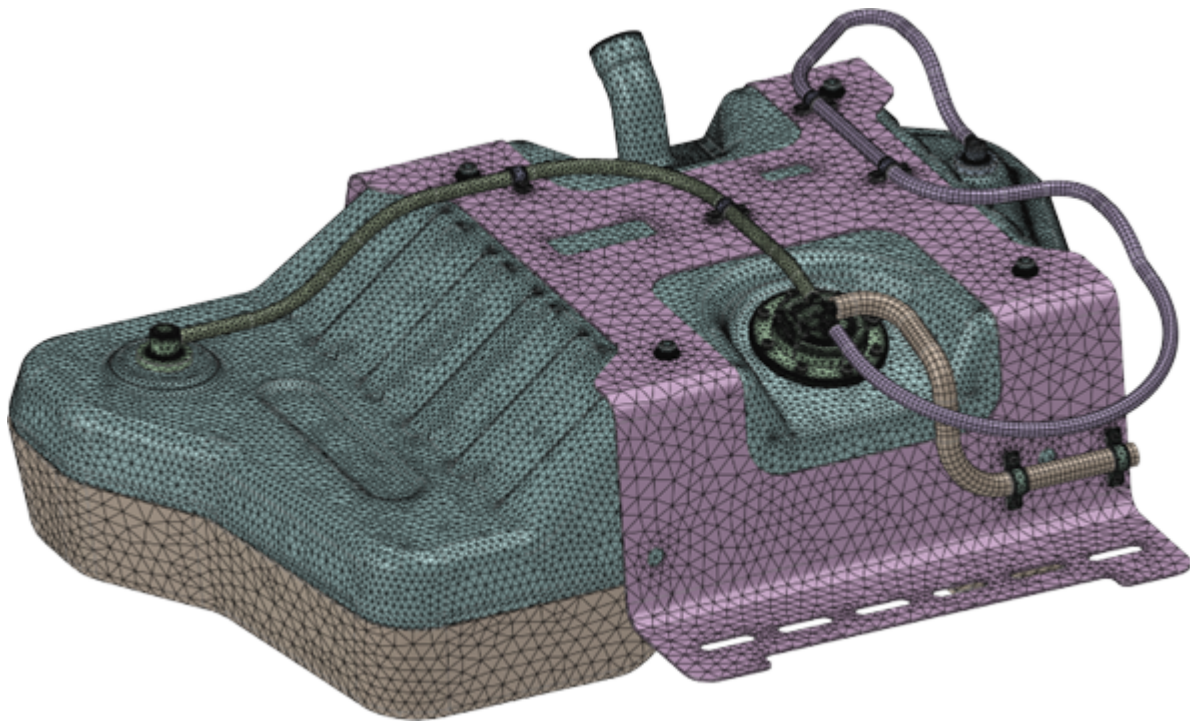
2.7.3. Improved Structural Meshing Defaults (Refine)

The structural meshing default settings in **Refine** have been enhanced to use the curvature and proximity size function with improved element sizes. The new default settings improve meshing robustness for a wide variety of model geometry and enhance mesh quality for thin solid structures.

Default Mesh in 2025 R1

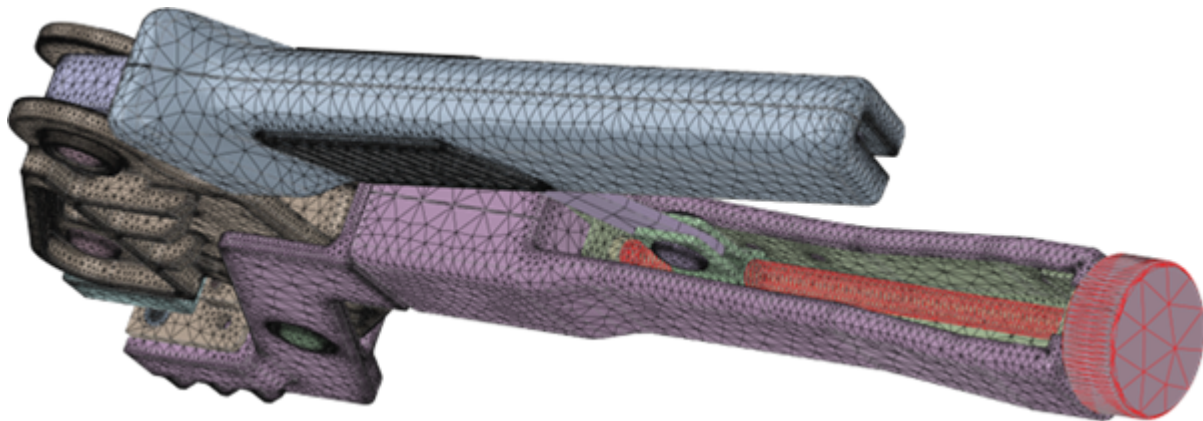


Default Mesh in 2025 R2



2.7.4. Highlighted Failed Tetrahedral Mesh (Refine)

The Discovery application now highlights any failed mesh in red. This helps you identify problematic geometry for meshing in **Refine**. This feature is available for all tetrahedral meshing.



2.8. Modeling Tools

Note:

Many modeling tools are common between the Discovery and SpaceClaim applications, although implementations may differ depending on product.

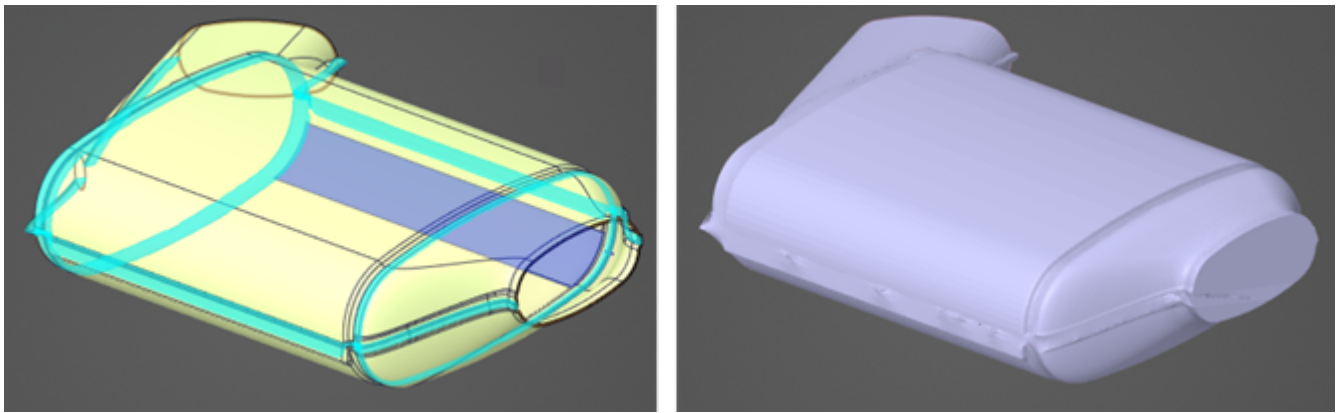
2.8.1. Improvements to Modeling Documentation

Additional modeling documentation that was originally developed for the SpaceClaim application is available in the Discovery Documentation's Table of Contents. This change provides easier access to detailed conceptual and reference information about modeling functionality.

You may notice anomalies in the modeling documentation when accessed from the Discovery application. Improvements are ongoing.

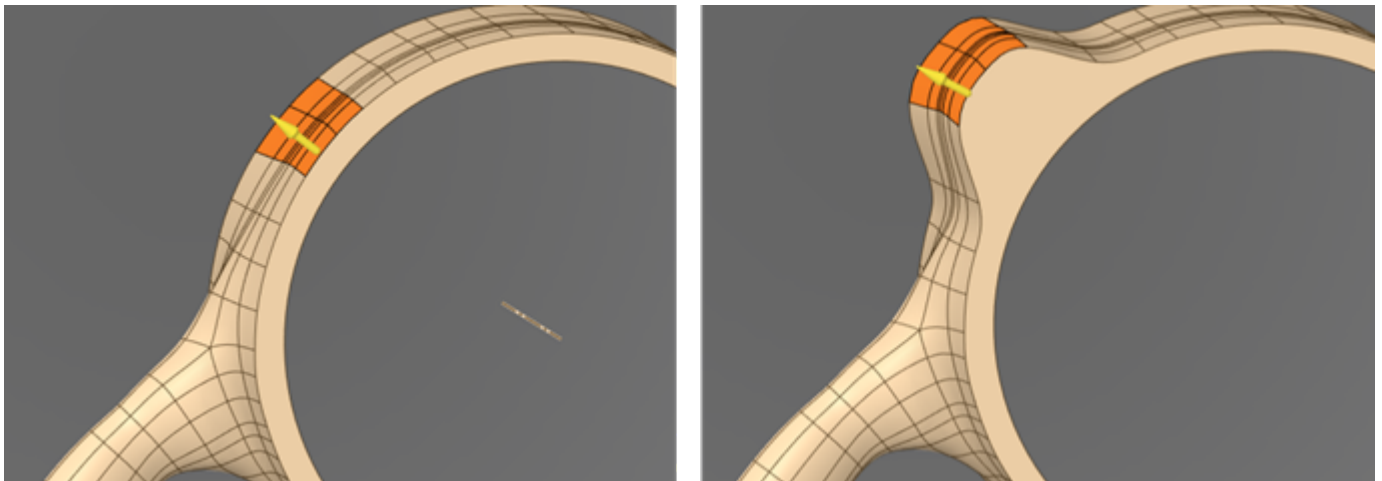
2.8.2. Tolerant Volume Extract

You can now extract the facet-based tolerant volume when there are gaps between the bodies in the model using the **Extract tolerant volume** option in the **Volume Extract** tool. The facet-based tolerant volume is extracted for bodies where the gaps are smaller than the **Maximum gap** size specified.



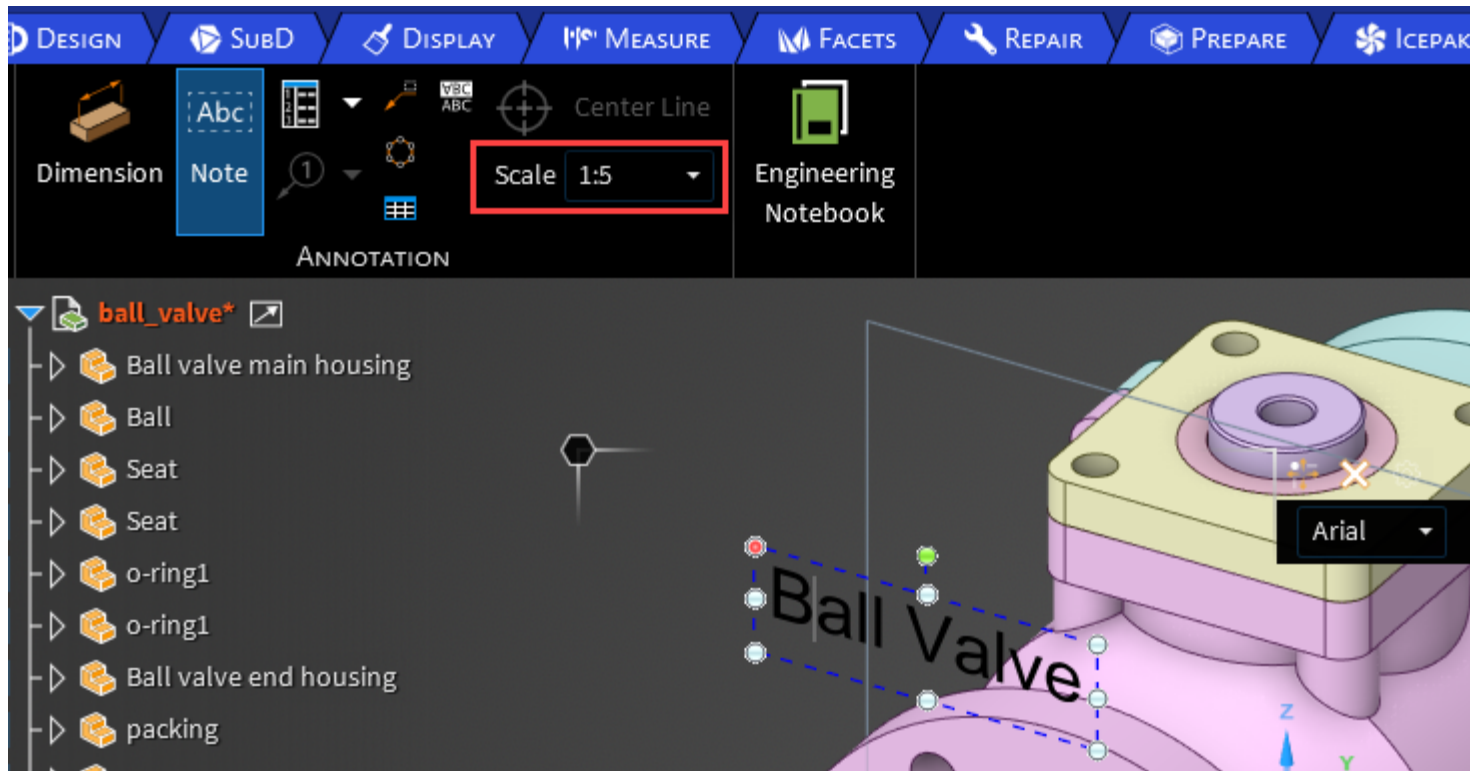
2.8.3. Improvements to Hybrid Modeling

You can now use **Move** or **Pull** to edit the edges and/or vertices that lie on the boundary between SubD regions and traditional surfaces such as planes or cylinders. The edges and vertices will remain constrained to the traditional surfaces/curves which they border. Note that because of the different modeler approaches, the boundary edges may become tolerant as they attempt to match both SubD and traditional geometry.



2.9. Text Scale for 3D Annotations

You can now configure the text scale for 3D annotations. This setting applies uniformly to all annotations, scales proportionally with the model, and updates in real time.



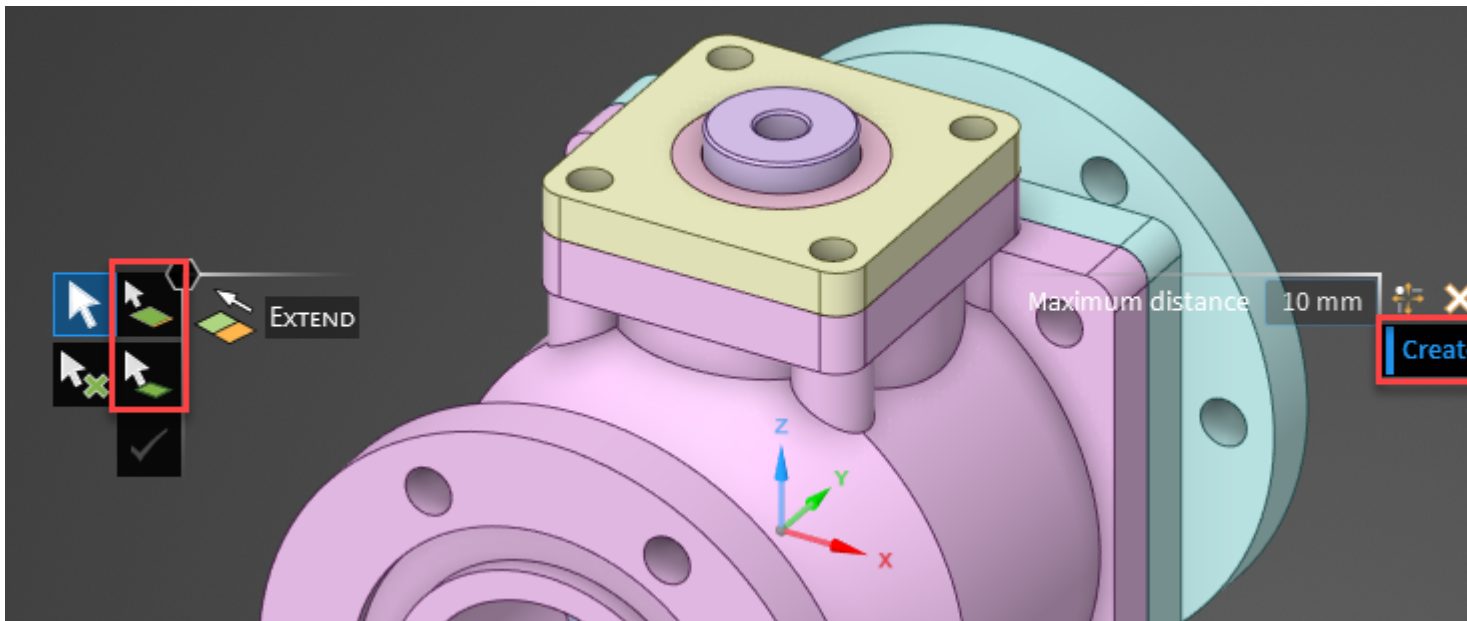
2.10. Extend Tool Updated

The **Extend Tool** now includes tool guides to assist with creating weld bodies between selected geometry. These guides appear when **Create body** is selected and provide contextual instructions for usage:

Create body creates a weld body between two selected edges or faces.

- **Select faces** - Choose the faces where the weld body will be created.
- **Select edges** - Choose the edges where the weld body will be created.

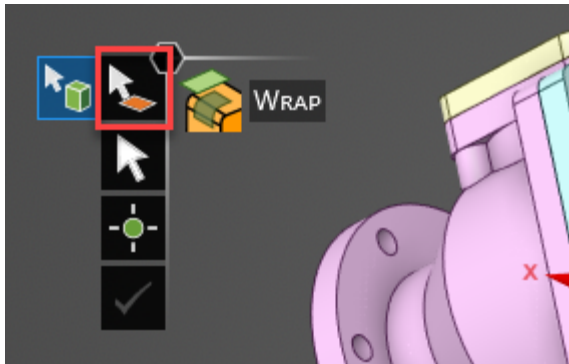
This enhancement improves usability by guiding users through valid selection workflows for weld body creation.



2.11. Wrap Tool Updated

A new tool guide has been added to the **Wrap Tool**. Apply **Restrict wrapping** to a selected set of target faces. This limits the wrap operation to only the faces you select, providing greater control over where the wrap is applied.

This enhancement improves precision when wrapping geometry to complex surfaces.



2.12. Materials

2.12.1. Materials Data for Simulation

Materials Data for Simulation (MDS) is available through an optional license that provides access to simulation-ready materials data for a broad coverage of materials classes, enabling engineers to create more accurate simulations with trusted data. For easy access, MDS is embedded directly into the Discovery application and currently contains 777 materials covering a wide range of material classes. The MDS materials represent a subset of the much larger material set available in Ansys Granta products.

Material Class	Count
Ceramics	47
Composites	37
Fluids	37
Foams	16
Glasses	15
Honeycombs	3
Magnets	27
Metals - ferrous	150
Metals - non-ferrous	229
Polymers	182
Woods	34
Total	777

Developments have focused on continuing to provide accurate and reliable simulation-ready materials data with standard updates to point and curve data.

Six new records have been added for the 2025 R2 release:

- Rubber, silicone (thermal interface material)
- Plastic, ABS+PC (15% stainless steel fiber, flame retarded)
- Plastic, ABS (15-20% stainless steel fiber)
- Plastic, PA66 (20% carbon fiber)
- Plastic, silicone molding compound (50-70% glass fiber)
- Plastic, bisphenol epoxy molding compound (mineral filler)

Additionally, some data values have been updated.

2.12.2. Materials Data for Simulation - Sample Library

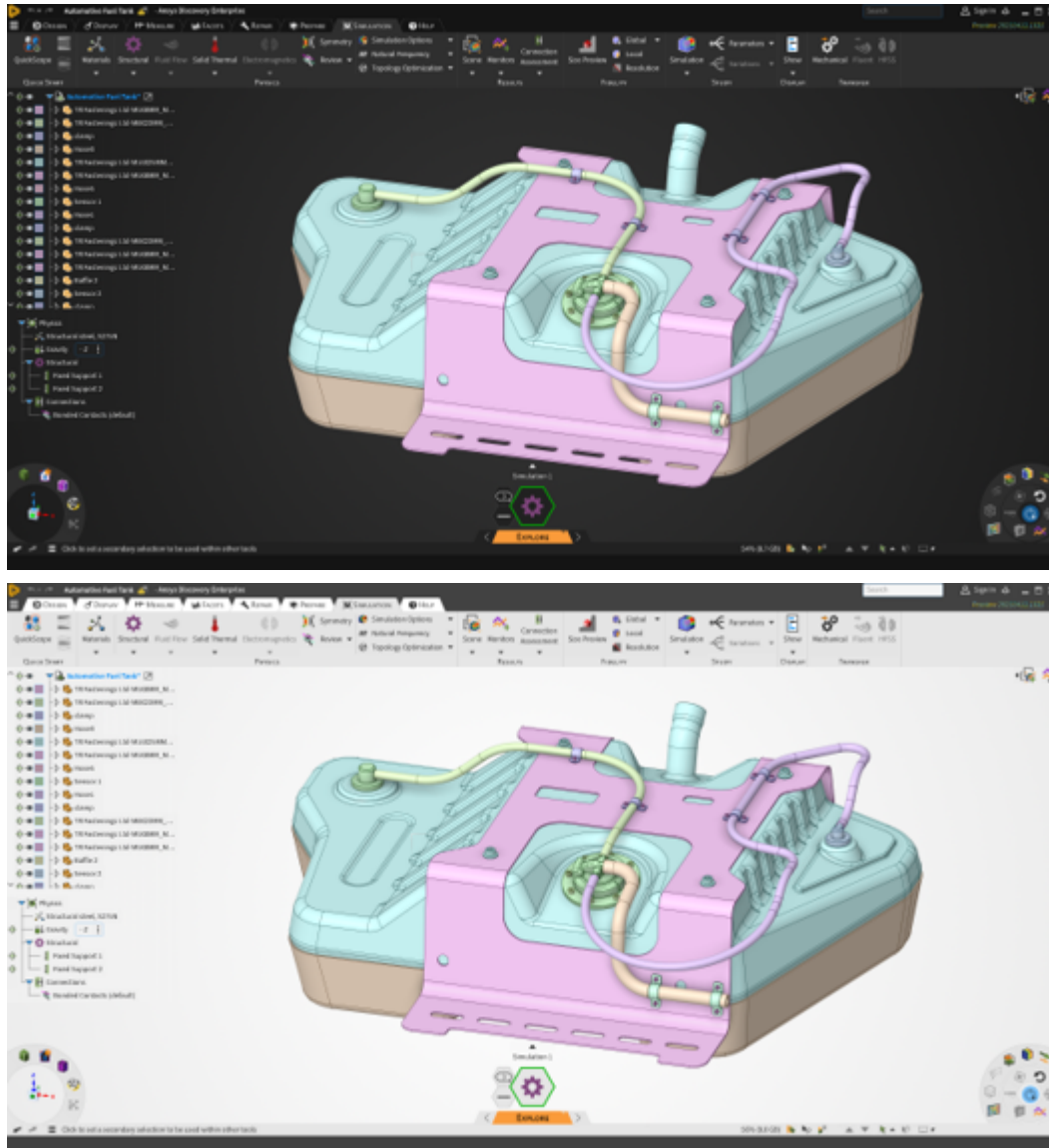
The Materials Data for Simulation (MDS) Sample library provides 39 materials free of charge to the Discovery application users with basic room temperature point data for mechanical and thermal properties. The MDS materials represent a subset of the much larger material set available in Ansys Granta products.

There are no new records in the 2025 R2 release of MDS, but some data values have been updated.

2.13. User Interface Enhancements

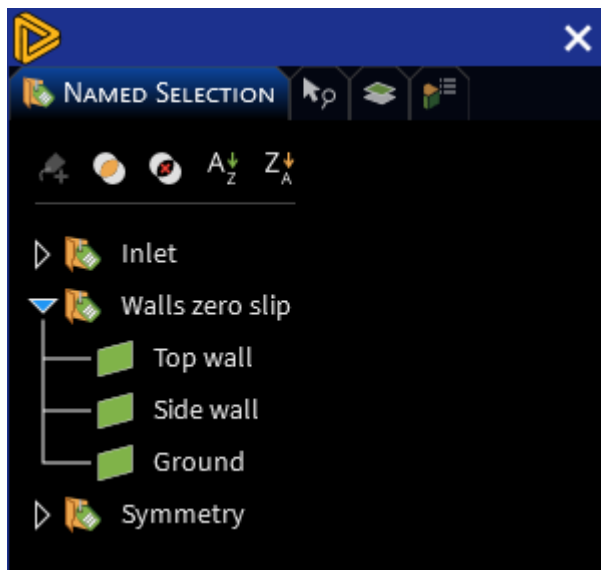
2.13.1. Preview the New UI Theme

Under **Settings > General > User Interface**, a **Preview new theme colors** option offers new, enhanced versions of the classic **Dark** (current default) and **Light** UI themes. These new themes prioritize the visual acuity of the model and results, standardize color use, improve contrast, and resolve past inconsistencies.



2.13.2. Grouping Named Selections

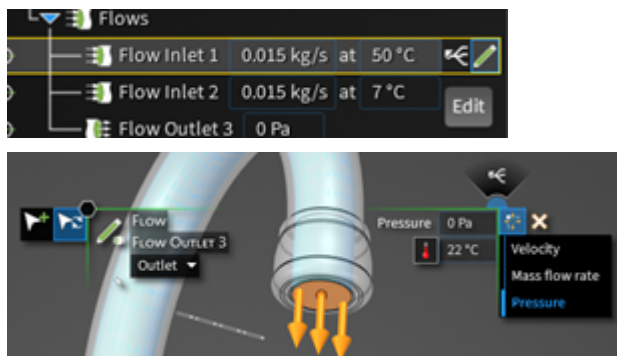
You can now organize named selections by defining folders to group the named selections for quicker access.



Select named selections from the list, right-click and select **Create Folder** from the context menu to group the named selections. Use the **Ungroup** option to ungroup previously grouped named selections.

2.13.3. Editing Physics Conditions

An improved workflow for editing physics conditions simplifies the task and feels more consistent with other HUD behavior. To edit an existing physics condition from the tree, hover over the condition and click the pencil or double-click the condition. Either action opens a condition-specific HUD and the options displayed are narrowed to those relevant to the condition. This reduced HUD makes complex modifications easier including reassigning locations or changing input values.



2.13.4. Assign Workflow Consistency

The user interfaces for the Assignment tools (Bolts, Beams, & Materials) have been updated and standardized for ease of use.

2.14. User Settings Summary

2.14.1. Fluent Solver Settings

The following **Fluent Solver** options have been added under **Solver Settings** in the **Physics** section.

File Menu > Settings > Physics > Solver Settings

- Select **Enable Fluent solver option** to allow you to use the LiveGX solver or turn it off. If you turn off **Use LiveGX solver** under **Simulation Options Defaults (Refine)** for fluid flow, then the Fluent solver will be used in the simulation.
- Select **Transfer to Fluent (restart required)** and then choose **Export cas.h5 file** (default) to use this file format for transfer to the Fluent application. When the **Export journal file** option is selected, a journal file is saved along with the mesh and settings file and can be later imported in the Fluent application.

2.14.2. Reference Pressure Default

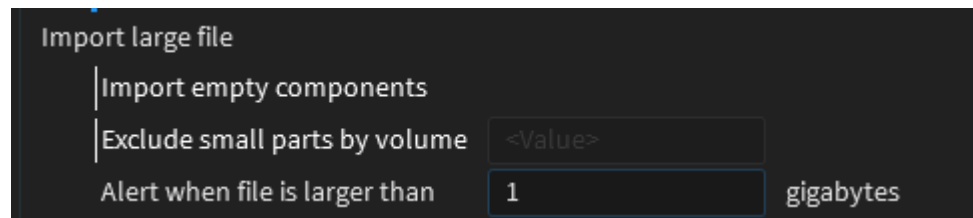
Reference pressure is a new option under **Simulation Options**. The reference pressure you specify is used to adjust the calculation of the force on a wall. You can also set a default **Reference pressure** in **File Menu > Settings > Physics** under **Simulation Options Defaults (Refine)** for fluid flow.

2.14.3. Import Large File Options

When importing large files, you can now configure **Import large file** behavior using new options available under **Settings > File > General**, in the **Import** section:

- **Import empty components**- Choose whether to include empty components in the import process.
- **Exclude small parts by volume** - Specify a volume threshold to automatically exclude parts smaller than the given value.
- **Alert when file is larger than <value> gigabytes** - Enable a notification when the file size exceeds the specified limit (default: 1 GB).

These options help streamline large file imports by reducing unnecessary geometry and improving performance.



2.14.4. Recalculate Rendering

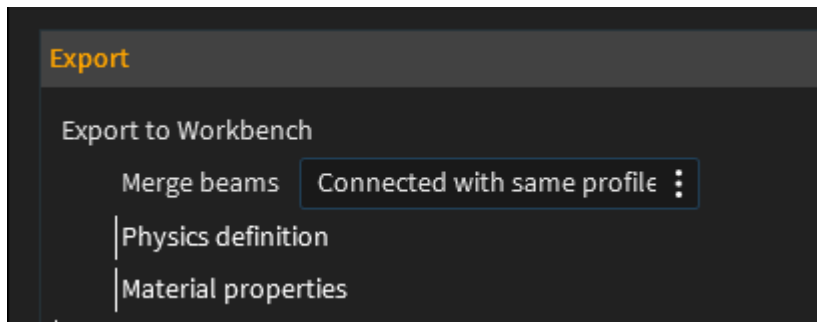
Use this option to manually refresh the 3D display in the graphics window after making custom changes. In certain cases, visual updates may not occur immediately due to rendering performance optimizations. Although the underlying geometry remains accurate, the display may appear misaligned or rendered at reduced quality.

2.14.5. Export to the Workbench Application

The following options are now supported when exporting to the **Workbench** application:

- **Merge beams** - Choose how beam bodies are grouped during export:

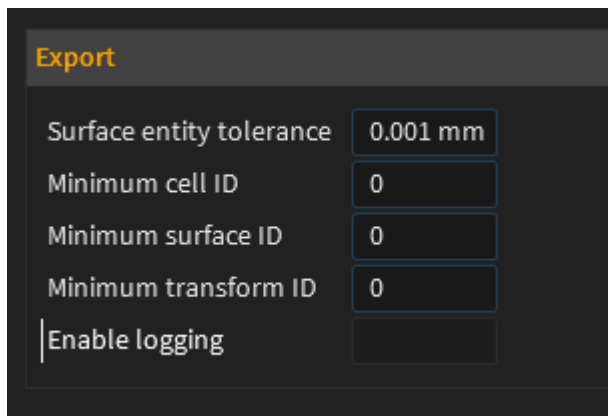
- **Never**- No merging occurs. Each beam remains a separate part in **Workbench**, regardless of connectivity or profile.
- **Connected with same profile**- Merges beams only if they are physically connected (share a node) and have the same cross-sectional profile. This reduces part count while retaining meaningful divisions.
- **Disconnected with same profile** - Merges all beams with the same cross-sectional profile, even if they are not physically connected. Use this option to further simplify the model by grouping similar beams.
- **Physics definition** - Select this option to export physics conditions, solver settings, and other simulation setup data to downstream solvers.
- **Material properties** - Choose to export material assignments and related simulation data.



2.14.6. New Export Support: MCNP (Monte Carlo N-Particle)

Export functionality has been added for **MCNP** (Monte Carlo N-Particle) format. The following export options are available:

- **Surface entity tolerance** - Defines the minimum allowable distance between surface entities to treat them as distinct. This helps control precision and prevent unintended merges during export.
- **Minimum cell ID** - Sets the starting index for assigning cell IDs in the exported MCNP file.
- **Minimum surface ID** - Sets the starting index for assigning surface IDs.
- **Minimum transform ID** - Sets the starting index for assigning transform IDs used in geometric transformations.
- **Enable logging** - Enables export process logging to the specified output file, useful for diagnostics and validation.



2.15. Results

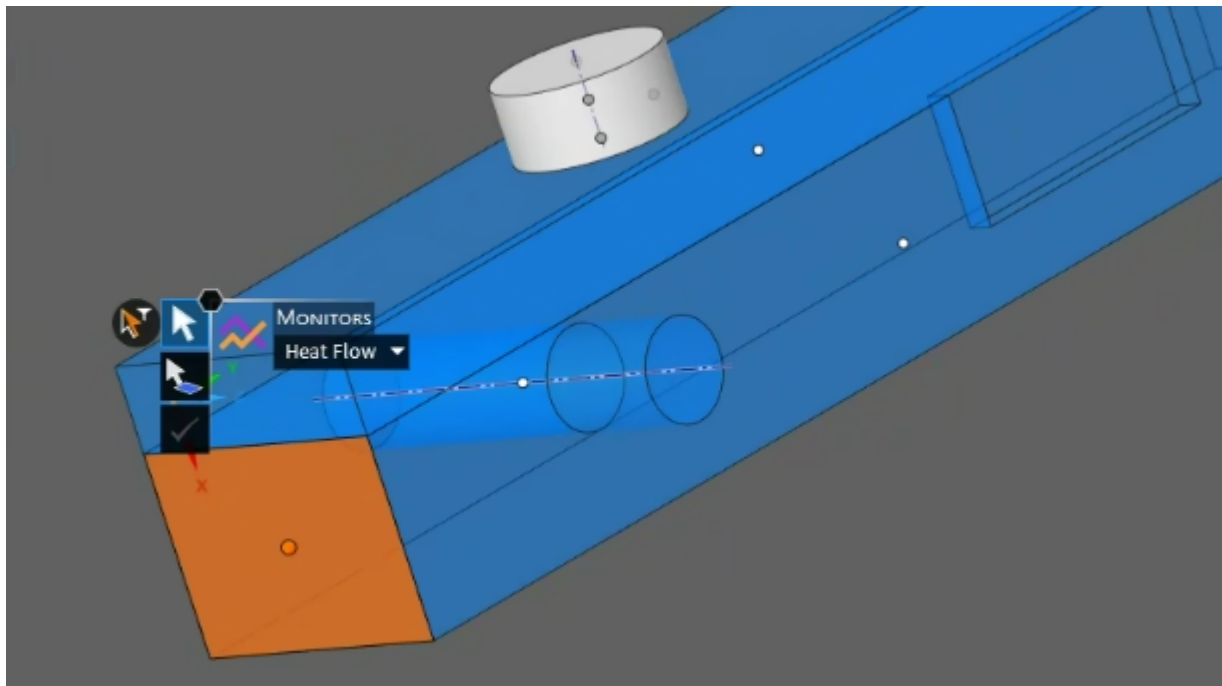
2.15.1. Save Animation to File

Saved scenes has expanded functionality to save the current animation to a *.mp4 file. Access **Save Animation to File** through the **Scene** dropdown list in the Simulation tab. This includes support for:

- Animation of deformation and modal deformation in **Explore**.
- Animation of deformation, modal deformation, streamlines, and curved vectors in **Refine**.

2.15.2. Heat Flow Monitors in Explore

Heat flow monitors are now available in **Explore** for flow inlets and outlets, walls, and fluid-solid interfaces.

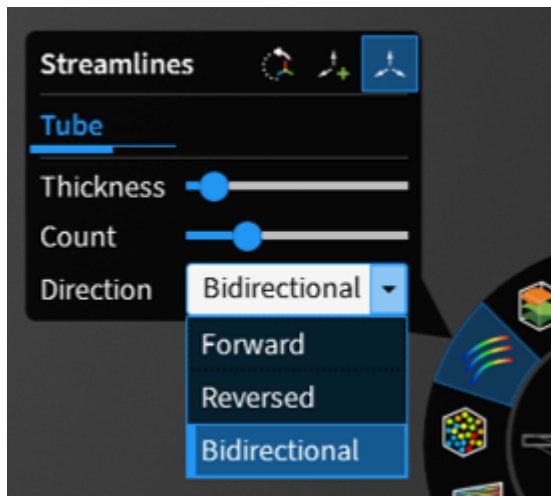


2.15.3. Force Monitors on Fluid-Solid Interfaces

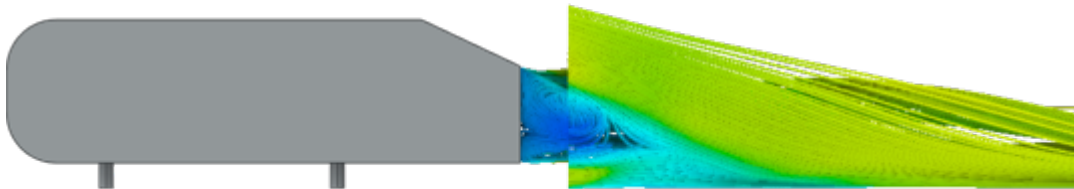
Force monitors are now available for fluid-solid interfaces. The results are available only on the fluid side.

2.15.4. Streamlines

For emitters that are not located at the inlet, you can now specify the direction of the streamlines to be forward and reversed by selecting **Bidirectional**.



Forward Streamlines

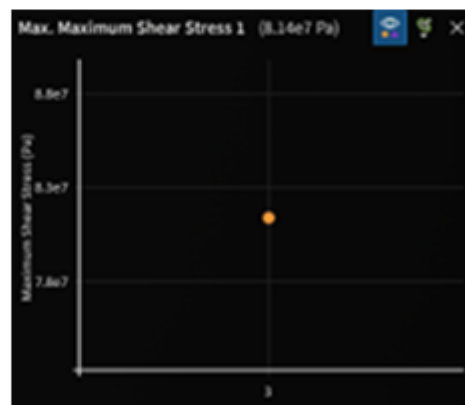
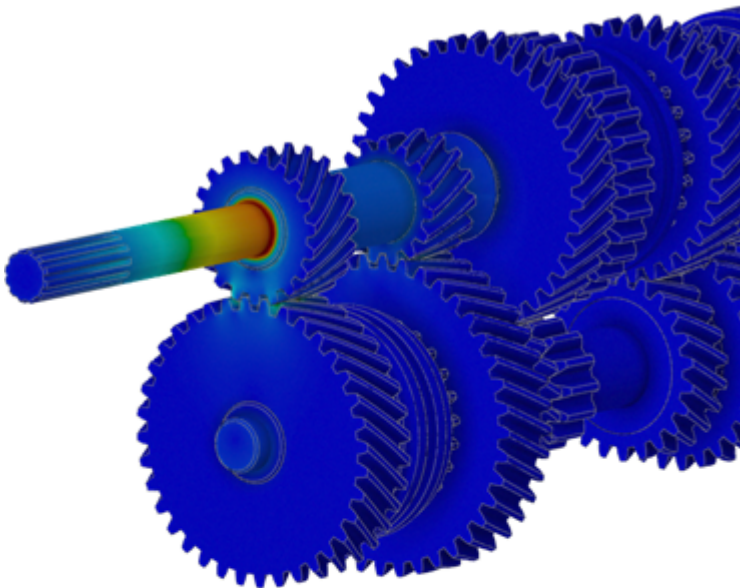


Bidirectional Streamlines



2.15.5. Max Shear Stress (Tresca)

The Discovery application now supports monitors for Max shear stress (Tresca) for all Structural simulations. This result is useful for evaluating structural components transferring torque loads, drive shafts, camshafts, etc.



2.15.6. Moment Reaction on Supports

The Discovery application now supports monitors for Moment reactions on supports, displacements and symmetry. This feature helps in evaluation of static equilibrium.

2.15.7. Selection of Conditions for Reaction Monitors

You can now select support or displacement conditions in the Physics tree for reaction monitors.

2.15.8. Reaction Force

The **Reaction Resultant** result has been renamed to **Reaction Force**. This result is single value.

2.16. Performance

Application performance and modeling responsiveness remains a top priority in this release. You may notice performance improvements in all stages from refresh rates to solution rendering, including:

- Better overall performance and fewer unresponsive
- Smoother interactions with the model
- Improved stability when running in a remote desktop (RDP) configuration

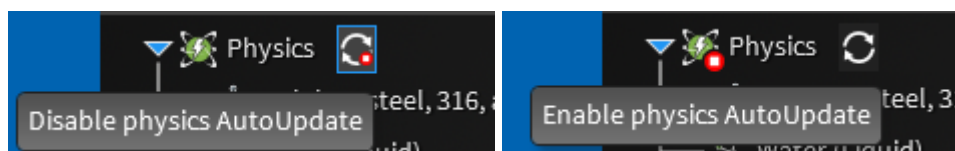
2.16.1. Enhanced Graphics Frame Rate and Remote Desktop (RDP) Fix

Enhancements to how the application handles graphics frame rate have improved overall performance, including smoother interactions with the model. By using a different approach to integrating 3D and 2D UI, this may alleviate or resolve:

- Known performance issues related to running in a configuration with remote desktop (RDP)
- Some initialization issues with configurations involving multiple GPUs where the application can fail to launch due to there being more than one GPU that supports the application rendering
- General performance navigating 3D scenes using the mouse, space-mouse or keyboard

2.16.2. Physics AutoUpdate

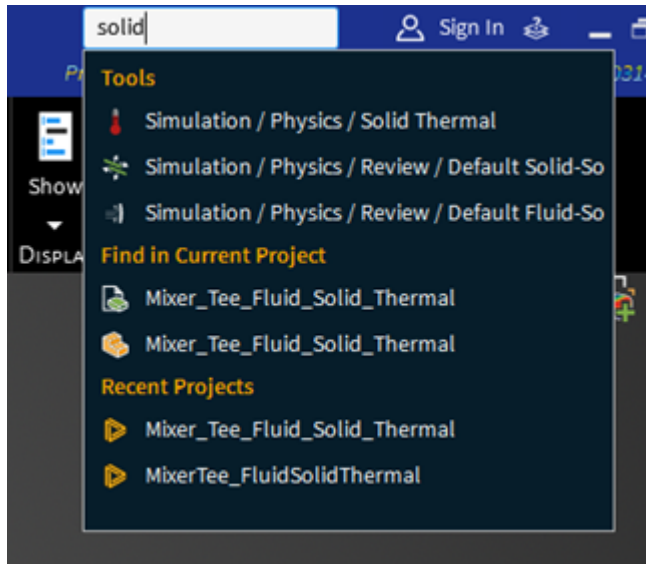
Physics AutoUpdate updates the physics tree as you make modifications to the model, keeping the model and physics trees in synch. To improve performance when making model changes, you can pause **Physics AutoUpdate** by selecting the **AutoUpdate** icon at the top of the tree and then turn it back on once model changes are complete to resynch the trees.



2.17. In-Product Guidance

2.17.1. Search

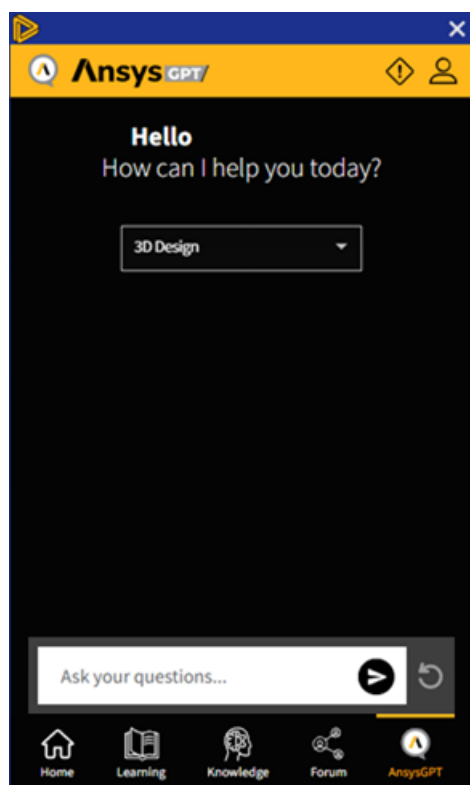
The new search feature allows you to search for tools in the active application, objects in the current project, and project files in your recent projects list.



2.17.2. Help Resources

A new **Help** tab is available, which contains links to online resources. With an AnsysGPT AI+ license increment, **Ansys Engineering Copilot** is available in the **Help** tab. **Ansys Engineering Copilot** opens a window where you can access several resources that will help you find answers to your questions and learn something new by engaging with the large global Ansys community. Additionally, you can use **AnsysGPT™** to provide answers to your questions pertaining to your application.

In the absence of an AnsysGPT AI+ license increment, **Ansys Innovation Space** is available in the **Help** tab.



2.18. Installation and Licensing

Use of the [Defined Variations Generator \(p. 11\)](#) to create defined variations automatically based on an algorithm requires the optislang_level1 or dsdxm licensing increment. All three license levels (Pro, Premium, and Enterprise) began including the dsdxm licensing increment in early 2025. If you are using an older license key and want to use this functionality, you may need to request a new key.

2.19. Ansys 3D Design Verification Manual

A new fluids test case was added: *Laminar Flow in a 90° T-Junction*.

3. Discovery Application Caveats and Known Issues

At the time of publication, caveats and known issues in Release 2025 R2 of the Discovery application are listed below. Also refer to the *Ansys, Inc. Known Issues and Limitations* document.

- The Discovery application may become unstable or crash after hibernation or reconfiguring hardware such as, but not limited to, multiple monitors or docking or undocking a laptop. **Workaround:** Check the log and confirm the existence of the following exception:

ID: System.Runtime.InteropServices.COMException

Title: COMException

Message: UCEERR_RENDERTHREADFAILURE (Exception from HRESULT: 0x88980406)

Go to the **Settings > General > Graphics Performance** and turn on **Enhance graphics frame rate (restart required)**. [1205135]

- When using the performance option **Enhance graphic frame rate** in **Settings > General**, drag and drop in the UI is not supported. **Workaround:** Use **File > Open** to open or insert a file. [1247931]
- When using the performance option **Enhance graphic frame rate** in **Settings > General**, the in-product, interactive help (overlay) is not supported. **Workaround:** This is a software limitation and there is no workaround. [1251147]
- Once the Discovery application is installed, opening Excel will prompt to install an addin. **Workaround:** Select **Don't Install** when prompted by Microsoft Office Customization Installer. [1263635]

SpaceClaim Release Notes

Release 2025 R2 of SpaceClaim contains the following.

- 1. [SpaceClaim Product Advisories](#)
- 2. [SpaceClaim Changes Overview](#)

1. SpaceClaim Product Advisories

SpaceClaim users should be aware of the advisories described here.

SpaceClaim Geometry Kernel

- The underlying geometry kernel for SpaceClaim changed from ACIS to Parasolid at Release 2025 R1.
- SpaceClaim Release 2024 R2 was the last version based on ACIS.

Installation

The SpaceClaim application was removed from the Ansys unified installers and the standalone Discovery application installer at Release 2025 R1, but you can still install it using the methods described in [Installation \(p. 33\)](#).

API Version Support

API version V23 has been deprecated at Release 2025 R2 and will be removed at Release 2026 R1.

If you have customized add-ins or scripts, make sure they are upgraded to the most current version.

2. SpaceClaim Changes Overview

The following changes have been made at Release 2025 R2:

- 2.1. [Installation](#)
- 2.2. [Scripting](#)
- 2.3. [Import/Export Support](#)

2.1. Installation

Although the SpaceClaim product is no longer included in either the Discovery product installation or in the **Ansys, Inc. Installation Launcher** (Ansys unified installer), you may still install SpaceClaim if needed. Use one of these methods to install SpaceClaim:

Downloading the Installation Package From the Customer Portal

You can download the SpaceClaim installation package from the Ansys Download Center and install it:

1. Sign in to your account. If you don't have an account, you are prompted to create one.
2. From the list of **Downloads** on the left, select **Current Release**.
3. Expand **Add-On Packages** and locate the **SpaceClaim** section. Click **Full Package** to begin the download.
4. From the Windows File Explorer **Downloads** folder, extract the .zip file to a convenient location on your computer.
5. In the folder of extracted files, double-click **setup.exe** and follow the prompts to install SpaceClaim.

From the Command Line

When running the Ansys unified product installation from the command line, you can include the **-install_spaceclaim** option to install SpaceClaim.

For example: **setup.exe -install_spaceclaim**

Obtaining SpaceClaim as a Component of Another Product

When you install Material Designer or Speos, SpaceClaim is also automatically installed and configured.

2.2. Scripting

- The current Scripting version is v252.
- API version V23 has been deprecated at Release 2025 R2 and will be removed at Release 2026 R1.

If you have customized add-ins or scripts, make sure they are upgraded to the most current version.

2.3. Import/Export Support

- The legacy SpaceClaim *.sdoc files are imported using a translator. Opening these files may take longer due to the geometry conversion.
- The saving of legacy SpaceClaim *.sdoc files is no longer supported. You can only save files in the *.sdocx format.

DesignModeler Release Notes

1. DesignModeler Product Advisories

DesignModeler users should be aware of the advisories and product changes described here.

2. DesignModeler Enhancements

DesignModeler 2025 R2 has no new features or enhancements.

3. DesignModeler Caveats and Known Issues

At the time of publication, there are no caveats or known issues in Release 2025 R2 of DesignModeler. Also refer to the *Ansys, Inc. Known Issues and Limitations* document.

Ansys Structural Products

Licensing

At Release 2025 R2, Ansys Mechanical products have transitioned to the new Ansys Common Licensing model. Additional license increments have been added. The previous bundled license increments that enable multiple applications (for example, ansys, mech2, mech1) remain unchanged. For more information, see [Summary of New and Changed Features in the Ansys, Inc. License Management Guide](#).

Release notes are available for the following Ansys Structural products:

[Mechanical Application \(p. 39\)](#)

[Mechanical APDL \(p. 57\)](#)

[Autodyn \(p. 77\)](#)

[Aqwa \(p. 79\)](#)

[ACP \(p. 81\)](#)

[Material Designer \(p. 85\)](#)

[Additive Suite \(p. 87\)](#)

[Sherlock \(p. 91\)](#)

[Forming \(p. 99\)](#)

[Motion \(p. 103\)](#)

[LS-DYNA \(p. 105\)](#)

Mechanical Application Release Notes

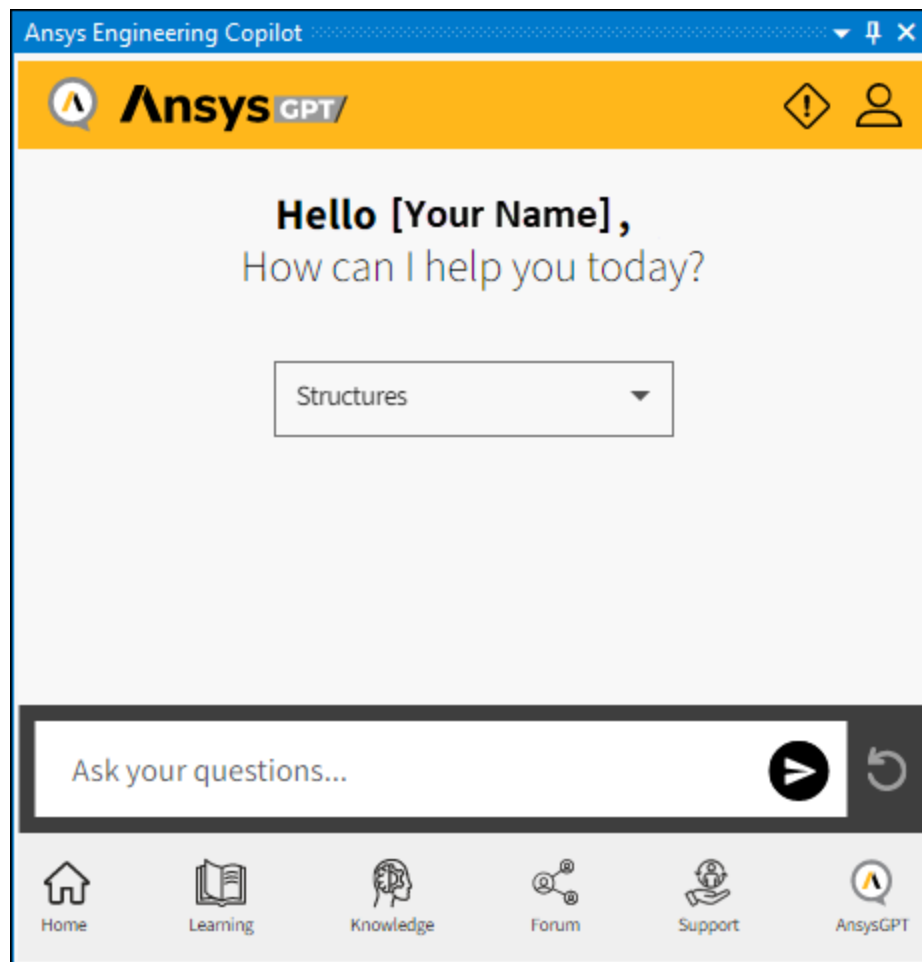
This release of Ansys Mechanical™ contains all of the capabilities from previous releases plus many new features and enhancements. Areas where you will find changes and new capabilities include the following:

1. Ansys Engineering Copilot
2. Changes in Product Behavior
3. Performance
4. Add-ons
5. General
6. Graphics
7. Geometry
8. External Model
9. Contact and Connections
10. Mesh
11. Additive Manufacturing Process Simulation
12. Analysis
13. Fracture
14. Substructure Analysis
15. Structural Optimization
16. Motion
17. Loads/Supports/Conditions
18. Imported Loading Conditions
19. Solution
20. Explicit Dynamics
21. Results
22. Scripting and Customization
23. Ansys GRANTA Materials Data for Simulation

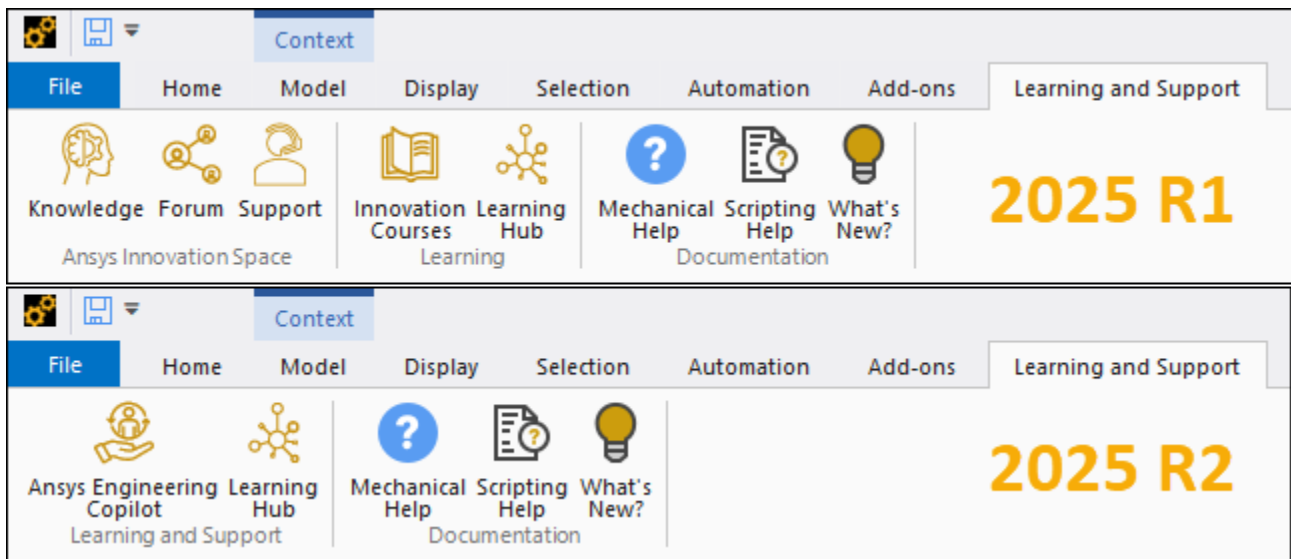
1. Ansys Engineering Copilot

The **Ansys Engineering Copilot**™ hub for learning and support is now available in the Mechanical application. This AI-powered feature integrates over 50 years of Ansys technical support and engineering expertise, putting it at your fingertips and accessible within seconds. Activated from the [Learning and Support](#) tab, this feature eliminates the need to go to individual external web sites. Instead, it provides a centralized location for the **Learning**, **Forum**, and **Knowledge** resources as well as direct access to

Ansyes training and the **AnsyesGPT™** virtual AI assistant. The Ansys Engineering Copilot™ feature is a licensed product available at no additional cost, but it requires the **AnsyesGPT AI+** license to use it.



With this implementation, the options in the Learning and Support tab have been updated. The Knowledge, Forum, and Innovation Courses options, previously displayed on the tab, are now integrated into the Ansys Engineering Copilot feature. This feature allows you to search for a specific topic and then provides search results that direct you to various training courses, forum topics, and more.



2. Changes in Product Behavior

Release 2025 R2 includes new features and enhancements that result in product behaviors that differ from previous releases. These behavior changes are presented below.

- **Input File Creation.** To reduce file size and improve the processing time required to write the input file (ds.dat), new formats have been implemented for the following two commands:
 - **CMBLOCK:** For the new format, the entities (element/node IDs) that form a sequence of more than two consecutive numbers are written by specifying the first ID followed by last ID of that sequence with a hyphen.
 - **EBLOCK:** This command now uses the COMPACT format where the first field is the element ID, and the remaining fields are the node IDs. Other fields for material, element type, real constant, section number and element coordinate system are specified just before the **EBLOCK** command.

See the [Coded Database File Commands](#) section of the *Mechanical APDL Programmer's Reference* for more information about these commands.

- **Imported Condensed Part.** Previously, for MSUP Harmonic systems, the application, by default, applied **Imported Condensed Part** load vectors during the Use pass with a scale factor of one (1.0). Now, instead of the application automatically applying load vectors, you need to apply and scale these load vectors using the [Load Application](#) object during the Use pass.
- **Contact Region Advanced Settings.** The **Projected, Uncoupled U to ROT** option of the [Constraint Type](#) property has been renamed **Mixed Assembly**.
- **Coriolis Effect.** For a harmonic-based analysis when the **Solution Method** property is set to **Full**, if you set the **Coriolis Effect (Analysis Settings > Rotordynamics Controls)** property to **On, Rotating Reference Frame**, the application no longer considers centrifugal force. In previous releases, the application considered this force a harmonic load.
- **Remote Boundary Conditions.** Now, when you scope a remote boundary condition to the edges of line body that are collinear, the application supports the **Deformable** setting for the **Behavior**

property. Previously for this setting, the application internally changed the setting to **Rigid** during the solution process.

- **Structural Optimization - Absolute Maximum Approximation.** Previously, during a **Structural Optimization** analysis that included a displacement **criterion** specified for **Absolute Maximum** Spatial Reduction, regardless of method, the application used an approximation for the maximum value. Now, for certain methods, Mixable Density, Level Set, Shape Optimization, and Topography, which have a displacement-based criterion on a static linear model, the application has been refactored in order to produce a displacement value that is much closer to the actual maximum value.
- **Connection Group:** The **Faces** option of the **Group By** property now merges multiple contacts which have the same scoping on the target side and overlapping scoping on the contact side. In earlier releases application would keep them as individual contacts.
- **Automatic Mesh Update after Body and Joint Configuration.** The application now automatically updates the mesh following both body and joint configuration actions, when either the **Set** or **Revert** option is used. Previously, mesh updates had to be performed manually after these configurations.
- **Pre-Meshed Crack.** The Mechanical APDL command written to the input file for the crack extension direction definition of a **Pre-Meshed Crack** has changed. The application now writes the command **CINT,EDIR** instead of the assist node-based extension direction specification done through the **CINT,CTNC** command.
- **Accelerated Animation.** Now, when you export a result animation, the application may choose to select the **Accelerated Animation** setting. Previously, when you exported an animation, the application always used the **Standard Animation** setting. This change may enable the application to produce the video faster.
- **Contact Tool Initial Information.** The default display of the **Possible Overconstraint** conditions in the initial contact information worksheet has changed. Now, for this condition, the column displays by default when the **Formulation** property is set to either **Program Controlled** or **MPC** and the **Type** property is set to **Bonded** or **[Bonded, Initial]**, otherwise it is not available. Previously, the information displayed regardless of the contact type setting.
- **Pasting Data into the Tabular Data Window.** The ability to copy data from a Microsoft ExcelTM spreadsheet and paste it into the **Tabular Data** window has been enhanced. Previously, clipboard data was truncated when copying high-precision scientific numbers. Now, the application copies all underlying values of the cells as well as metadata, ensuring greater accuracy without data loss or conversion errors.
- The **Export CAD File** option for a Motion analysis is no longer available, following migration to the new Parasolid graphics kernel.

3. Performance

The following performance related enhancements were implemented for Release 2025 R2:

- **Input File Creation.** Enhancements have been made to improve the performance of input file creation. The application now implements compact formats to write the **CMBLOCK** and **EBLOCK** commands. As a result:

- The input file is now generated significantly faster. For some larger models, it has been observed that this process can be up to 45% faster than the previous release.
- The generated file is dramatically smaller in size. This is especially true for large models, for which the input file size may be up to 67% smaller compared to the previous release.
- **Importing NASTRAN Files using External Model** . Now, when importing NASTRAN files from [External Model](#), the application parses files faster, but more significantly, the time it takes to load finite element file data into Mechanical is as much as 60% faster compared to previous releases.

4. Add-ons

The following updates were made to the Mechanical add-ons for Release 2025 R2:

- The **DesignLife** add-on offers the following capabilities:
 - The ability to [Filter Duplicated Mesh IDs](#) with higher damage or lower safety factor.
 - Spot Weld [Analysis Settings](#) for **Weld Diameter** and **Method**.
 - Support for both independent and mesh-dependent beams in a [Spot Weld](#) analysis.
 - A [Mean Stress Correction](#) setting has been added for **Spot Weld** analysis.
 - [Life and Damage Vector](#) results are supported for a **Solid Seam Weld** analysis.
 - The WeldHotSpot [Solution Location](#) case is supported for a **Stress Life** analysis.
 - The ability to read standard SN/EN curves from a [Materials Database](#) for **Stress, Safety Factor** and **Strain Gauge** analyses.
 - The [PSD Table and Curve](#) are now integrated into the Mechanical application display.
 - [Complex Algorithm](#) properties are exposed for all the **Frequency Domain Loads**.
 - You can now use the [Commands](#) object to access properties or property values that are not exposed in the DesignLife add-on, but which you wish to modify in the `input.dcl`.
 - **Young's Modulus** and **Tensile Ultimate Strength** properties must now only be defined in the [Engineering Data](#) for those materials used in the model.
 - All properties passed to the `input.dcl` created by the DesignLife add-on are now written in alphabetical order, for easier debugging and comparison.
 - For **Solid Seam Weld** analysis, the `solid_seamweld_config.xmls` is now generated automatically. The [Solid Weld Auto](#) property allows you to reuse this file on subsequent runs.
- The **NVH Toolkit** add-on now includes support for:
 - Multiple output nodes in the [FRF Calculator](#) and improvements in UI and performance.
 - User-defined [Frequency Spacing](#) for the **FRF Calculator**.

- Rigid body modes for the [FRF Calculator](#).
- UI enhancements for [Mode Contribution](#), including mode specification for table plot and bar chart, simultaneous Mode Contribution and FRF plots, and improved bar chart sorting.
- Improved [Pre-Test Calculator](#) performance.
- The **Forced Response** add-on has been updated to include the following features:
 - A [Modal Force](#) result, which displays the content of the `fileFSUMT.matr` matrix file in tabular data and graphical format.
 - A [Mode Multiplier](#) plot, which displays the content of the `fileaVec.matr` matrix file in tabular data and graphical format.
 - The capability to generate [Cold Coordinate](#) temperature and pressure maps.
 - The [Multiharmonic Plot](#) has additional stress and strain results for Shell and Solid nodes and elements.
 - The [Multiharmonic Chart](#) has additional stress results for Shell and Solid nodes and elements.
- The **Rotordynamics** add-on has been expanded to include:
 - [Critical Speed Map](#) support for up to 3 bearings.
- A new **Custom Results** add-on has been created to enhance result interpretation and validation using [SEPC](#) and [Relative Deformation](#).
- Updates for the **Additive Manufacturing** add-ons are described in the Additive Manufacturing Release Notes.

5. General

The following general enhancements were made at Release 2025 R2:

- **Store Results At Preference.** You can now change the default setting of the **Store Results At** property (**Analysis Settings > Output Controls**) using the new **Store Results At** preference contained in the **Output Controls (Static and Transient)** category of the [Analysis Settings and Solution](#) category of the [Options](#) dialog.
- **Invert Visibility:** This option now has an associated hotkey combination for its use: **Ctrl+I**. From either the Geometry window or an object in the Outline, this feature enables you to hide all currently displayed bodies and display all currently hidden bodies.
- **Automatically Backup Mechanical Projects.** When you open Mechanical *independently*, there is a new [Options](#) dialog preference, **Backup Project Before Save**. Active by default, this preference instructs the application to create a backup folder of all simulation files prior to saving the simulation to make sure that you do not lose any data in the event of an error during the process of saving your project. This preference is available in the new dialog category **Project Management**.

6. Graphics

The following graphical display enhancements were made at Release 2025 R2:

- **Creating Coordinate-Based Section Planes.** Now, when you create a **Section Plane** on a given Cartesian coordinate system, you can choose the desired plane upon which to create the **Section Plane**, either the **XY Plane**, **YZ Plane**, or **XZ Plane**. In previous releases, the application default, and only option, was the XY Plane.
- **Show Hatch Lines.** The **Section Plane** window now includes a new display option: **Show Hatch Lines**. When you have capped faces displayed (**Show Capping Faces**), this option enables you to toggle the display of the hatch (black parallel) lines shown on the section plane on or off.
- **Anti-Aliasing (MSAA) Preference.** The **Anti-Aliasing (MSAA)** display preference, that previously only improved the display of a model in the **Geometry** window, has been refactored to support a variety of additional display options, including image exports, animation exports, print preview, and report preview. Furthermore, this capability now supports the Linux platform for all of the above display options.

7. Geometry

The following geometry enhancements were made at Release 2025 R2:

- **Importing ECAD Geometry into Mechanical.**
 - When you open Mechanical independently, the application now supports the ability to import ECAD (electronic computer-aided design) geometries in order to perform an ECAD analysis. Supported file types include *.def, *.tgz, and *.cvg.
 - **ECAD Import** now supports the Linux platform.
- **Body Merge** now supports **Virtual Topology** under the following conditions:
 - You can add a **Body Merge** object if your model includes a **Virtual Topology**.
 - Body merge fails if you try to merge bodies that share faces, edges, or vertices that are part of a **Virtual Topology**.
 - You cannot add a **Virtual Topology** object if your model includes a **Body Merge** object.

8. External Model

The following **External Model** system enhancements were made at Release 2025 R2:

- **Processing Improvements for Nastran.** When importing Nastran files from **External Model**, the application now parses files faster, but more significantly, the time it takes to load finite element file data into Mechanical is as much as 60% faster compared to previous releases.
- **External Model Imported Object Worksheet.** New data types are now available for query using the **Mechanical Scripting** window. You can now query worksheets for lists of IDs for the following imported objects:

- Imported Contact
- Imported Flexible Connectors
- Imported Rigid Connectors
- Imported Shell Thickness

See the [Static Structural Analysis using External Model](#) section of the *Scripting in Mechanical Guide* for examples of these queries.

9. Contact and Connections

The following contact and connection enhancements were made at Release 2025 R2:

Contact Enhancements

- **Contact Region.** The **Contact Region** object has two new options for the **Type** property:
 - **Bonded, Initial:** For this setting, the application models the bonded contact but maintains the contact detection for the initial state throughout the analysis. For a closed initial state, then contact remains attached during the analysis. For an open initial state, contact remains open, even if deformation brings the contact pair close together.
 - **No Separation, Penetration Only:** For this setting, separation of the geometries or gap is not allowed between the contact and target surfaces. The contact is established only when the surfaces are either touching or penetrated.
- **Modeling Option for Edge Contacts.** The **Contact Region** object has a new [scoping property](#): **Modeling Option**. This property is displayed for edge contacts when the [Edge Contact Type](#) property is set to **Nodes on Edge** or when the contact is scoped to a [General Axisymmetric](#) body. The **Modeling Option** displays the read-only value **Force Based**, that indicates that the contact condition is using the contact force-based computational model.
- **Contact Region Advanced Settings.** The **Projected, Uncoupled U to ROT** option of the [Constraint Type](#) property has been renamed **Mixed Assembly**. The functionality of the option remains the same.

Connections Enhancements

- **Coincident Length Factor.** A new property, **Coincident Length Factor**, is now available for [remote points](#) and certain [remote boundary conditions](#) when you set the **Behavior** property to **Beam**. You use this property to specify a factor to adjust the location of an internally generated node that the application creates in order to avoid a zero-length beam element. This application generated node affects the stiffness of the connection with the remote point when a mesh node and the pilot node of the remote point have coincident locations. Supported remote boundary conditions include:
 - Point Mass
 - Remote Points
 - Joints

- Springs
- Beams
- Bearing
- **General Joint Type.** The **Rotations** property for the **General** joint type has three new options: **Free X and Free Y**, **Free Y and Free Z**, and **Free X and Free Z**. These options enhance your ability to further constrain the rotational degrees of freedom of your model. With these new options, the application now enables you to constrain a body under rotation for all six degrees of freedom.
- **Connections Worksheet.** The **Connections Worksheet** now provides filtering options. You use these options to specify the data displayed in the tables for the available Connections Information (Contact, Joint, Spring, Beam). You can activate and/or deactivate filtering options using the available check boxes or using the context (right-click) menu.

10. Mesh

For Release 2025 R2, the following mesh enhancements have been made:

- **Imported Point Mass.** When you import a mesh-based geometry from a CDB source file that includes **Point Masses**, the application now supports the **MASS21** element with **KEYOPT(1) = 1**.

For all other mesh related enhancements, refer to the [Meshing](#) release notes.

11. Additive Manufacturing Process Simulation

See [Additive Manufacturing](#).

12. Analysis

Release 2025 R2 includes the following analysis enhancements:

- **Load Vector and Base Excitation Application in MSUP Harmonic Analyses.**

You can now specify the following loading conditions in a [Modal Analysis](#):

- [Acceleration](#) and **Acceleration** defined as a **Base Excitation**
- [Displacement](#) defined as a Base Excitation
- [Pressure](#)
- [Force](#)

These loads do not affect natural frequencies or mode shapes. However, you can use the base excitation and load vector data from these boundary conditions and apply and scale them using the new **Load Application** object to a downstream Mode Superposition (MSUP) Harmonic analysis.

This new loading condition, **Load Application**, enables you to take base excitations or load vectors created by an upstream **Modal** analysis or imported from an **Imported Condensed Part** object and apply and scale them in a Mode Superposition (MSUP) Harmonic analysis.

Also note that previously, for MSUP Harmonic systems, **Imported Condensed Part** load vectors were applied during the Use pass with a scale factor of one (1.0) by default. Now, instead of automatic application of load vectors, you apply and scale these load vectors using the **Load Application** object during the Use pass.

- **Additional Property Support for Modal Analyses.** The following properties are now included in a **Modal** analysis:
 - **Include Residual Vector:** Available in the **Options** category of the **Analysis Settings** object. Property options include **Yes** and **No** (default). When set to **Yes**, the application calculates residual vectors for the **Modal** analysis. You can use these generated residual vectors in a downstream **Harmonic Response** MSUP analysis when that analysis also has the **Include Residual Vector** property set to **Yes**.
 - **Load Control:** Available in the **Pre-Stress** object if you also have an upstream **Static Structural** analysis specified for the **Modal** system. You use this property to define load generation for prestressed analyses.
- **Random Vibration.** The Random Vibration analysis now supports a new loading condition: **PSD Pressure**. This load provides pressure in terms of spectral value vs. frequency based on your choice of the pressures that were applied in the prerequisite modal analysis.
- **Campbell Diagram Support.** You can now create a **Campbell Diagram** from a prestressed **Modal** analysis that includes a **Bearing** that is defined using the **Tabular Data** function that is dependent on the specified Rotational Velocity.
- **Solution Interrupt.** The **Progress Pane Interrupt** option is now supported during **Coupled Field Static** and **Coupled Field Transient** analyses.
- **Analyses Selection.** The **Analyses Selection** feature now enables you to activate or deactivate **Point Mass** and **Distributed Mass** objects for each analysis in your simulation.
- **Analysis Settings > Output Controls.** The **Output Selection** property of the **Output Controls** category of the **Analysis Settings** object has two new options:
 - **On Mode File By Named Selection.** For displacement results (only), during Modal and Harmonic Response analysis that have on demand expansion active, this option enables you to filter the content of the mode file and only write the results for the nodes of the selected named selections.
 - **Follow Upstream Analysis:** This setting is available when you have a **Harmonic Response** or **Transient Structural** analysis linked to an upstream **Modal** analysis. The application defaults to this setting and automatically uses the **Output Controls** settings defined in the upstream **Modal** analysis when your upstream **Modal** analysis has the **On Demand Expansion Option** property set to **Yes**.
- The **Motion Load** add-on for an **RBD** analysis now allows you to export the initial assembly configuration to a **Static Structural** subsystem and choose multiple transient time steps from which to read the loads.

13. Fracture

Release 2025 R2 includes the following [Fracture](#) analysis enhancements:

- **SMART Crack Growth.** You can now use the **SMART Crack Growth** feature to solve mixed mode fracture problems by including the Mode III Stress Intensity Factor- SIFS (K3) in Equivalent SIF calculations using the new **Equivalent SIF Method** and **Kink Angle Method** properties. This capability requires that the **SIFS** property (**Analysis Settings** > [Fracture Controls](#)) is set to **Yes**. This is the default setting.
- **Fracture Object.** When performing a [Fracture](#) analysis, you can now convert the local hex-dominant mesh to a quadratic tetrahedral mesh *within* the crack buffer zone using the following new options of the **Fracture** object property, **Re-mesh Hex-dominant to Tetrahedral**. These options retain the hex-dominant mesh outside of the crack buffer zone.
 - **Buffer Zone - MPC Contact:** Use this option to create a bonded MPC contact pair at the mesh interface between the hex-dominant elements and the tetrahedrons.
 - **Buffer Zone - Pyramid Elements:** Use this option to create pyramid elements at the mesh interface between the hex-dominant elements and the tetrahedrons.

Note that these options only support the fracture parameters calculation - not **SMART Crack Growth**.

14. Substructure Analysis

Release 2025 R2 includes the following [Substructure Analysis](#) enhancements:

- **Imported Condensed Parts.** Now, when you are working with the **Worksheet** of an **Imported Condensed Part** during a [Substructure Analysis](#):
 - And you create a Joint from the list of available interfaces, the application now automatically reassociates the Joint's original scoping when you re-import the **Imported Condensed Part** after you have cleared the generated data of the object.
 - A new **Contact Region** option is available in the context (right-click) menu. Use this option to create a Contact Region from one of the available interfaces listed in the worksheet.
- **Substructure Analysis.** From the **Workbench** Project Schematic, you can now link the **Solution** cell of a [Substructure Generation](#) analysis to the **Model** cell of a supported downstream system to automatically create an [Imported Condensed Part](#) object in the downstream system. This process automatically generates two [Exported Condensed Part](#) (*.cpa) files and enables you to select the file type, either the **Use Pass** or **Use Pass** and the **Expansion Pass**, to be used to create the **Imported Condensed Part** in the downstream system.

15. Structural Optimization

Release 2025 R2 includes the following [Structural Optimization](#) analysis enhancements:

- **Optimization Methods.** A new optimization method is now available: **Materials Optimization**. This method enables you to determine the optimal distribution of different materials in one domain.
- **Member Size Manufacturing Constraint.** The **Member Size** manufacturing constraint has been enhanced. The **Minimum**, **Maximum**, and **Gap Size** properties have been reconfigured into separate categories and a new **Measurement Type** property now enables you to further define how maximum thickness and gap size are computed. Using the properties associated with the **Measurement Type** property, you can now define the thickness and gap size on a selected plane of the model based on a specified coordinate system.
- **User Defined Criteria.**
 - For a **Steady-State Thermal** analysis, you can now specify **User Defined Criteria** using the **Primary Criterion** object (only). This feature includes the base result **Temperature** to evaluate thermal responses. You use this data to quantify the performance of your design or you can use it in a downstream **Structural Optimization** analysis to specify the **Objective** object or for a **Response Constraint**.
 - The **Spatial Reduction** property has a new option: **Standard Deviation**. This option computes the standard deviation of the base result over the selection.
- **Local Design Restriction.** For the **Shape Optimization** and **Topography** methods only, the **Type** property of the **Local Design Restriction** object has the following new options:
 - **Along Direction Morphing:** The application only displaces the nodes of the specified selection along a defined coordinate direction.
 - **Along Normal Morphing:** Nodes of the selection can only be displaced along the initial normal of the surface.
 - **Free:** The application can displace nodes in any direction up to a specified distance.
 - **Inward Morphing:** The application can only displace nodes inward up to a specified distance.
 - **Outward Morphing:** The application can only displace nodes outward up to a specified distance.
- **Pull Out Direction Manufacturing Constraint.** Now, for the **Topology Optimization - Level Set Based** optimization method, you specify a **Pull Out Direction** constraint that has the **Direction** property set to **Both Directions**, a new category of Details pane properties is available: **Parting Surface Control Group**. The category includes the property **Parting Surface Control**. Using this property, you specify whether or not to impose a parting surface and if so, the definition of this parting surface.

16. Motion

The following enhancements were made to Motion at Release 2025 R2:

- A **Create All Geometry** feature has been added to simplify and speed up CAD generation for Shaft, Housing and Drivetrain Assembler components.

- Internal performance improvements have been implemented, greatly reducing modeling times for Planetary Gearset and E-Axle components.
- A new graphics API has been implemented to improve view control performance while an entity is selected and highlighted. This affects the following objects:
 - **General:** Curveset, Curve Contact Properties, and EHD Bearing.
 - **Toolkits:** Motor EM force, Link Assembler, Path Follower, and Tire.
- The [Links Toolkit](#) has been updated to display real segment geometries when assembling the links system.
- A contact [Gap Detection](#) property has been added to control whether the solver calculates the gap distance during analysis.
- An option has been added to save simulation results after [Co-simulation](#) using the Functional Mock-up Interface (FMI).

For changes to the Motion solver, see the [Motion Standalone Release Notes](#).

17. Loads/Supports/Conditions

The following loads/supports/conditions enhancements were made at Release 2025 R2:

- **PSD Pressure.** For [Random Vibration](#) analyses, you can now specify **PSD Pressure**. This load provides pressure in terms of spectral value vs. frequency based on your choice of the pressures that were applied in the prerequisite modal analysis.
- **Heat Flow.** The **Heat Flow** loading condition now supports vertex selection when using the **Tabular** option to define **Magnitude**.

18. Imported Loading Conditions

The following imported loading condition enhancements were made at Release 2025 R2:

- **Load Mapping for Imported Body Temperature.** The **Mapped Data** property is now supported for the [Imported Body Temperature](#) load during the Thermal Stress analysis. This property provides the **To Binary File** option that uses an internal binary file to perform load mapping as compared to the legacy behavior that uses an ASCII file to map data using the **To Input File** option. This feature is also supported for import using **MAPDL Results File** option of the [Imported Load \(Group\)](#) object as well as when using [External Data](#).
- **Imported CFD Pressure.**
 - **Application Generated Named Selections for Imported CFD Pressure.** Now, when you import an [Imported CFD Pressure](#), the properties of the [Named Selection Creation](#) category of the Details pane support the ability to automatically create element face-based Named Selections, for Unmapped, Mapped, and Outside nodes.
 - **Shared Reference Body.** Additionally, when you scope the **Imported CFD Pressure** to bodies that share faces or element faces, the **Shared Reference Body** property is now

available. This property enables you to select the body on which you wish to apply the pressure.

- **Automatically Create Velocities and Synchronize Analysis Settings for the H5DPF File Format.** When you import velocities from [External Data](#) using a Hierarchical Data Format (HDF) Version 5 (*.hdf5/*.h5) file, the imported data often contains numerous orders or frequencies, each with multiple RPM values. Now, using the context (right-click) menu option, **Create Velocities and Sync Analysis Settings**, from the [Imported Load \(Group\)](#) object, you can automatically synchronize the properties of your **Analysis Settings** object with imported velocity data.
- **Synchronizing Analysis Frequencies.** Now, when you import velocities from a Hierarchical Data Format (HDF) Version 5 (*.hdf5/*.h5) file using [External Data](#), or, from an upstream **Harmonic Response** analysis, a new context (right-click) menu option, **Sync Analysis Frequencies**, is available. This option enables you to synchronize the **Analysis Settings** based on the selected imported load's source RPMs and Frequencies. This eliminates the need to manually synchronize these settings.
- **Imported Velocity Source Dimension.** The application now supports the **Source Dimension** property for the [Imported Load \(Group\)](#) object when you import an [Imported Velocity](#) load from a Hierarchical Data Format (HDF) Version 5 (*.hdf5/*.h5) file using [External Data](#). This enables you to set the model dimension, **2D** or **3D**, of the source file.
- **Grouping Imported Loads in a Binary File.** The [Imported Remote Loads](#) object has a new **Grouping** property option: **To Binary File**. This option enables you to group all imported loads in a binary file. Once imported, the application creates a new object: **File Based Imported Loads**. Use the properties of this object to view load data by **RPM Selection**, **Remote Point**, and load **Type** in the **Graph** and **Tabular Data** windows. This is the preferred option if you are importing substantial amounts of load data as compared to directly importing loads which requires the application to generate a large number of objects.

19. Solution

Release 2025 R2 includes the following solution enhancements:

- **Resource Prediction.**
 - Now supports stand-alone Modal analyses using the **Iterative** setting for the [Solver Type](#) property. You can set the property manually or it is the default selection for the **Program Controlled** option. This setting enables you to estimate the required computational time, memory usage, and disk space for the analysis.
 - Prediction charts now display data for up to 128 cores.
 - Charts now utilize log scaling on the value axis for improved data representation and include an option to toggle between **log** (default) and **linear** scales.
- **Solution Interrupt.** The [Progress Pane Interrupt](#) option is now supported during **Coupled Field Static** and **Coupled Field Transient** analyses.

20. Explicit Dynamics

The Workbench application provides multiple systems that enable explicit solutions:

- Explicit Dynamics
- LS-DYNA

The enhancements for these systems for Release 2025 R2 are described below.

Explicit Dynamics

The following updates were made at Release 2025 R2:

- **Intel MPI upgrade.** The default version of Intel MPI used to run parallel Explicit Dynamics jobs has been upgraded to version 2021.14.2.

The Explicit Dynamic Analysis System is now in maintenance mode. Make sure to check the release notes with every release or contact your Account Manager or local support. More information may be found in the [Mechanical Beta Features](#) document.

LS-DYNA

For LS-DYNA systems, note the following enhancements for Release 2025 R2:

- **Battery Modeling.** The Randles model type, **Solid**, allows you to define a detailed battery with multiple components.
- **Electromagnetic ASCII results.** You can postprocess EM ASCII results using the Binout Tracker.
- **Electromagnetic results.** You can postprocess EM calculations in Mechanical and visualize the EM results (for electric voltage, total current density, Joule heat, and so forth).
- **Acoustics Improvements.** The LS-DYNA Acoustics system now supports the [Absorption Surface](#) and [Rigid Wall](#) loading conditions.
- **Expanded Materials Support.** Additional LS-DYNA [material models](#) have been made available in Engineering Data.
- **Import SPH Particles.** You can import SPH particles through External Model when available in the LS-DYNA input file.
- **Use New S-ALE Keywords.** The **Use New S-ALE Keywords** option is available for S-ALE simulations and allows you to use a more efficient version of the input file, which has new structured keywords and removes redundant keywords.
- **S-ALE Material Grouping.** The **Combine Materials** option allows you to group multiple parts that have the same material assignment from Engineering Data.
- **S-ALE Mesh Workflow.** The S-ALE Mesh Workflow supports the full range of meshing options available in the solver and loads the full mesh into Mechanical, allowing visualization at the pre-processing stage.

- **Tracers.** Tracers are available in the LS-DYNA system for ALE and S-ALE simulations in Mechanical. The Tracer adds a tracer particle, which saves the history of either a material point or a spatial point.
- **Post Processing Improvements.** The deformation and other contour plots are now displaying the Rigid Wall used in drop test or impact simulations.
- **Material Assignment Support.** The LS-DYNA system now supports [Material Assignment](#), for Bodies only.

21. Results

Release 2025 R2 includes the following result enhancements:

- **Line Chart.** For LS-DYNA analyses, the options and capabilities of the [Line Chart](#) feature have been significantly enhanced.

New **Chart Options** Window:

- **Expanded Operations and Filters:** More tools to manipulate and refine your data.
- **Display Options:** Customize how your data appears in the chart.
- **Chart Axes Properties:** Modify the properties of the chart axes for better visualization.

New Context (Right-Click) Menu Options:

- **Label Data:** Add labels to your data points for clarity.
- **Toggle Data Display:** Easily show or hide data included in the chart.
- **Delete Datasets:** Remove unwanted data sets directly from the chart.

These enhancements provide more flexibility and control over how you visualize and interact with your data.

- **Frequency Response Result.** For this result, you can now further define the stress-based plot of your graph based on the following new **Type** property settings:
 - **Equivalent Stress**
 - **Maximum Principle Stress**
 - **Middle Principle Stress**
 - **Minimum Principle Stress**
 - **Maximum Shear**
 - **Intensity**
- **Force and Moment Reaction Probes.** For **Force Reaction** and **Moment Reaction** probes:

- When you set the **Location Method** property to **Surface**, the associated **Extraction** property has a new option: **Select Only Nodes On Surface**. Using this option, the application only examines the nodes that lie on the specified plane, as defined by the [Surface](#) construction geometry.
- You can now scope a reaction probe to a [Body-Body Beam](#). This scoping requires you to set both the **Calculate Reactions** and **Nodal Forces** properties of the [Output Controls \(Analysis Settings\)](#) to **Yes**.
- **Mode Shape Contours for Total Deformation**. Now, when you set the **On Demand Expansion Option** property to **Yes** during a standalone [Harmonic Response](#) analysis, two new properties are available: [Modal Result](#) and [Mode](#). Setting the **Modal Result** property to **Yes** displays the **Mode** property that you use to specify a desired mode value that evaluated the result for mode shapes.

22. Scripting and Customization

The following enhancements to scripting and customization were made at Release 2025 R2:

- To keep Mechanical application state calculations efficient, re-computation of data model objects is sometimes suppressed. This can result in objects not being in their expected states after data model changes are made via scripting. This especially affects the callbacks associated with the following events in the Mechanical application:
 - Before geometry is updated
 - After geometry is updated
 - Before a solve starts
 - After a solve is finished
 - Before post processing is started
 - After post processing is finished

A new API call, `DataModel.RecomputeStates()` can be run after these callbacks have finished executing to recompute and update the states of the objects in the data model.

23. Ansys GRANTA Materials Data for Simulation

The following enhancements to Materials Data for Simulation (MDS) were made at Release 2025 R2.

Materials Data for Simulation (Mechanical, Discovery, Electronics Desktop, and Fluent)

Six new records were added:

- Rubber, silicone (thermal interface material)
- Plastic, ABS (15% stainless steel fiber, flame retarded)

- Plastic, ABS (15-20% stainless steel fiber)
- Plastic, PA66 (20% carbon fiber)
- Plastic, silicone molding compound (50-70% glass fiber)
- Plastic, bisphenol epoxy molding compound (mineral filler)

Additionally, some material data values were updated.

Materials Data for Simulation Sample Library (Mechanical, Discovery, and OnScale)

There are no new records, but some data values were updated.

Materials Data for Simulation Producer Library

Electronics Desktop

Standard updates have continued to provide accurate and reliable simulation ready materials data. There are 4 new records.

Motor-CAD

Standard updates have continued to provide accurate and reliable simulation ready materials data.

BH curves of several cobalt steel records are updated.

There are no new records.

Mechanical APDL Release Notes

Release 2025 R2 of the Mechanical APDL™ application offers most of the capabilities from prior releases plus many new features and enhancements. Areas where you will find updates and new capabilities include the following:

- [Structural \(p. 57\)](#)
- [Multiphysics \(p. 64\)](#)
- [Solvers \(p. 65\)](#)
- [Other Enhancements \(p. 66\)](#)
- [Commands \(p. 68\)](#)
- [Elements \(p. 70\)](#)
- [Documentation \(p. 72\)](#)
- [Technology Showcase: Example Problems \(p. 72\)](#)

For more information about this release, see the [Update Guide \(p. 72\)](#) section of this document and the [Global Release Notes](#).

1. Structural

Release 2025 R2 includes new features and enhancements for the following structural analysis disciplines:

- [1.1. Contact](#)
- [1.2. Elements and Nonlinear Technology](#)
- [1.3. Material and Fracture Modeling](#)
- [1.4. Linear Dynamics](#)
- [1.5. Transient Dynamics](#)

1.1. Contact

Release 2025 R2 includes the following new features and enhancements for analyses involving contact.

- [1.1.1. Arctangent Friction Model](#)
- [1.1.2. Enhanced Double-Sided Target Surfaces](#)
- [1.1.3. Enhancements to Multipoint Constraints and Assemblies](#)
- [1.1.4. Performance Improvements for MPC Bonded Contacts](#)
- [1.1.5. Enhanced Contact Pair and MPC Surface Based Constraint Generation](#)

1.1.6. Generate a Component from Rigid Target Elements

1.1.7. Using an Acronym with the RMODIF Command

1.1.1. Arctangent Friction Model

A new [frictional contact model](#) is now available using **TB, FRIC, TBOPT=ATAN**. This option models frictional stress as a continuous function of sliding velocity using an arctangent relation. Because the frictional stress is continuously differentiable with this option, it should help in non-linear problems that are having difficulty converging.

1.1.2. Enhanced Double-Sided Target Surfaces

In prior releases, the double-sided target surface option (KEYOPT(8) = 1 of the [TARGE170](#) element) was valid only for the 3D surface-to-surface contact element [CONTA174](#). In release 2025 R2, the double-sided target surface option is now valid for the 3D node-to-surface contact element [CONTA175](#).

Contact detection for the double-sided target surfaces is also refined to prevent missing constraint equations or bad crossing constraint equations for the MPC contact definition. This greatly improves contact solution accuracy and robustness.

For more information, see [Double-Sided Target Surfaces in the Contact Technology Guide](#).

1.1.3. Enhancements to Multipoint Constraints and Assemblies

Selecting a constraint type for assemblies is problem-dependent and can have a significant impact on solution accuracy. In release 2025 R2, the mixed constraint type option (KEYOPT(5) = 2 of target element) has been extended to the 3D node-to surface contact element [CONTAC175](#) and the 3D line-to-surface contact element [CONTA177](#) (in addition to the previously-supported 3D surface-to-surface contact element [CONTA177](#)). This change enables larger percentages of assembly models to be solved with improved robustness and accuracy, particularly for mixed constraint types (solid-to-solid, shell-solid, shell-to-shell) defined within a contact pair.

For more information, see the discussion of input data for [TARGE170](#).

1.1.4. Performance Improvements for MPC Bonded Contacts

Performance improvements for multipoint constraint (MPC) bonded contacts (CONTA element KEYOP(2)=2, KEYOP(12)=5,6) include the following:

- Better performance for MPC bonded contacts during static/transient assembly and mode expansion passes.
- Contact pair-splitting operations ([CNCHECK,DMP](#) and [CNCHECK,SPLIT](#)) produce more evenly distributed split MPC pairs across domains.

1.1.5. Enhanced Contact Pair and MPC Surface Based Constraint Generation

In most applications (especially larger assemblies), generating contact pairs and surface-based multipoint constraints (MPC) requires many lines of MAPDL script and multiple, repeated **EBLOCK** commands. This increases the size of input files and reduces their efficiency and readability.

To address this problem, release 2025 R2 includes two new commands:

- **CNTGEN** creates a contact pair based on element components with face ID (EFACE component(s) via the **CMBLOCK** command) or nodal components provided for the contact surface and the target surface. It generates contact elements **CONTA174** and target elements **TARGE170** over the given face of the exterior of a body meshed by 3D solid and shell elements.
- **SFCGEN** creates MPC surface based constraints based on element components with face ID (EFACE component(s) via the **CMBLOCK** command) or nodal components provided for the contact surface and the target surface. It generates one pilot node, a target segment meshed with the target elements **TARGE170**, and a contact surface meshed with either **CONTA174** or **CONTA175**.

By using these commands, you can create a contact pair or MPC surface based constraint with one MAPDL command and more efficiently define them in bigger assemblies. This also reduces the size of input files by eliminating the need to use multiple **EBLOCK** commands.

1.1.6. Generate a Component from Rigid Target Elements

You can now optionally create a new component from the rigid target elements generated by the **RBGEN** command. Use the *ECOMP* option to specify the name of a new component that includes the newly-generated elements. This makes it easier to list, plot, and debug rigid target elements.

1.1.7. Using an Acronym with the RMODIF Command

You can now optionally specify an acronym name instead of a serial number when you are using the **RMODIF** command to modify a real constant item for contact elements.

1.2. Elements and Nonlinear Technology

Release 2025 R2 includes the following enhancements to elements and nonlinear technology used in structural analyses:

- 1.2.1. Genc Joint Element
- 1.2.2. Joint damping support
- 1.2.3. Direct Element Embedding Enhancement
- 1.2.4. Mono Harmonic Option Support
- 1.2.5. Nonlocal Damage Formulation Support
- 1.2.6. Generalized Remeshing for Adaptive Nonlinear Damage (GRAND)
- 1.2.7. Nonlinear Adaptive Remesh Reasons
- 1.2.8. Adjusting Automatic Time Incrementation

1.2.1. Genc Joint Element

A new joint element is available. **MPC184-Genc** is a two-node joint element in which the relative rotations are characterized by a rotation axis. It is used to connect two bodies to transmit motion from one body to another. Stiffness and damping behavior can be specified for this joint.

1.2.2. Joint damping support

You can now activate joint damping behavior in a static analysis by setting `KEYOPT(6) = 1` for the relevant joint element. The relative velocities are computed based on incremental displacements and the time increment in the given substep. For more information, see [Stiffness and Damping Behavior of Joint Elements in the *Multibody Analysis Guide*](#) and the individual joint element descriptions.

To consider only the initial slope of a nonlinear stiffness or damping curve in a linear analysis, you can now use the option, `KEYOPT(8) = 1`, for the relevant joint element. See the individual joint element descriptions for more details.

1.2.3. Direct Element Embedding Enhancement

The [SHELL181/SHELL281](#) structural shell elements are now supported in the direct-embedding workflow. For more information, see [Direct-Embedding Workflow](#).

1.2.4. Mono Harmonic Option Support

The general axisymmetric solid elements, [SOLID272](#) and [SOLID273](#), now support the **MODE** command (using `KEYOPT(2) = 1`) and all nodal results can be saved to the `.rst` file (using `KEYOPT(3) = 1`). [SOLID272](#) with `KEYOPT(1) = 1` now supports the enhanced strain formulation.

1.2.5. Nonlocal Damage Formulation Support

[SOLID186](#) and [SOLID187](#) are now enabled to activate nonlocal damage models such as generalized damage (implicit gradient regularization). Nonlocal damage formulation capabilities can better capture the distributed nature of damage in real materials by averaging variables over a finite neighborhood and introducing a characteristic length scale. This leads to more objective and reliable results, improved convergence stability and mesh insensitivity in the presence of strain localization or softening.

1.2.6. Generalized Remeshing for Adaptive Nonlinear Damage (GRAND)

In the NLAD framework, the Generalized Remeshing for Adaptive Nonlinear Damage (GRAND) capability enables material removal by generating voids in an implicit analysis. The nonlinear adaptive criterion REMMAT is now used to initiate voids when high strains are reached during the solution process. See [Material-Removal-Based Criterion](#).

1.2.7. Nonlinear Adaptive Remesh Reasons

The REMESH REASON list in the remesh monitor file `.rmsh` now includes more remesh reasons and their combinations. Each remesh reason has one identifier number and when there is more than one reason, each identifier is listed in a row.

1.2.8. Adjusting Automatic Time Incrementation

You can now adjust automatic time incrementation to strictly adhere to the time increment specified by the **DELTIM** command. By default, automatic time incrementation adjusts the initial time increment to generate a whole number as the total number of substeps. To bypass this default, specify

ForceKey as an option for the **DELTIM** command, The program then adheres to the initial time increment that you specified.

1.3. Material and Fracture Modeling

Release 2025 R2 includes the following enhancements to material modeling and fracture analysis technology used in structural analyses:

- 1.3.1. Generalized Damage for Brittle Failure Modeling
- 1.3.2. AML Python Module Enhancements
- 1.3.3. Material Curve Fittings Enhancements
- 1.3.4. Small-deflection Hyperelasticity
- 1.3.5. Threshold SIF-range Saved as Result for SMART Fatigue Crack-growth
- 1.3.6. SMART Crack-Growth with Cyclic Symmetry Boundary Conditions
- 1.3.7. Enhancement of J-integral calculation

Some material properties are not available via the material property menus of the GUI. For a list of such material properties, see [GUI-Inaccessible Material Properties](#).

1.3.1. Generalized Damage for Brittle Failure Modeling

You can now apply a generalized gradient damage model to simulate a brittle failure problem (**TB,CDM,,,MXPS**). A maximum principal stress driven damage model is used to effectively capture a brittle rupture. It does not usually converge when simulating an unstable damage growth problem. Therefore, an appropriate small viscous effect can be considered to enhance the stability of the brittle damage simulation using the command **TB,CDM,,,VREG**. For details and an example, see [Regularized Generalized Damage](#).

1.3.2. AML Python Module Enhancements

The AML Python Module is the Python extension for Ansys Materials Lab (AML). It allows you to access Ansys structural materials to evaluate material tangent and stress for supported material models and perform parameter fitting for supported material models.

A new method, `auto_initialize`, is now available to perform AI-based initialization needed for the fitting process. The AML Python Module now supports several Extended Drucker-Prager and Geomechanics material models.

For more information, see [Using AML Python Module](#).

1.3.3. Material Curve Fittings Enhancements

Automatic coefficient initialization now supports the visco-hyperelasticity model by using a combination of hyperelasticity and Prony series. Temperature-dependent visco-hyperelasticity is supported for the combination of hyperelasticity, Prony series, and the shift function.

For more information, see [Visco-Hyperelasticity](#).

1.3.4. Small-deflection Hyperelasticity

Hyperelasticity is now supported with small-deflection formulation. Static, transient, modal, and harmonic analyses are all supported by small-deflection hyperelasticity. Small-deflection analysis uses the hyperelastic tangent modulus at zero deformation as an approximation to the more general finite strain hyperelasticity when the deflection is small.

1.3.5. Threshold SIF-range Saved as Result for SMART Fatigue Crack-growth

Threshold stress-intensity factor range (ΔK_{th}) specified for a SMART fatigue crack-growth analysis is stored as a result at all the substeps. It can be retrieved using **PRCINT**, ***GET** or **PLCINT** commands. The availability of the threshold range as a result is particularly useful for verification, when it is specified using a table or defined using a functional relation as in the NASGRO equations.

1.3.6. SMART Crack-Growth with Cyclic Symmetry Boundary Conditions

SMART crack-growth can be analyzed with cyclic symmetry boundary conditions in a single stage model. A crack can be initiated in the stage during solution, or a pre-existing crack can be used for the analysis. The cyclic symmetry constraint equations are established (using the **CECYCMS** command) for the matching node pairs on the low and high sector boundaries. If the crack growth remeshing affects the cyclic symmetry boundaries, the program automatically maintains a matching mesh on these boundaries and updates the constraint equations for the new mesh.

1.3.7. Enhancement of J-integral calculation

The Ramberg-Osgood model and kinematic hardening elastic-plastic material models are now supported in the calculation of J-integral for explicit cracks.

1.4. Linear Dynamics

Release 2025 R2 includes the following enhancements for structural analyses involving linear dynamics:

1.4.1. Rotordynamics Analysis

1.4.2. Static Multiframe Restarts for Cyclic Symmetry and Multistage Cyclic Symmetry Analyses

1.4.3. Node-based Mode File Scoping for Mode-Superposition Analysis

1.4.4. Python Script to Calculate Relevant Harmonic Indices for a Multiharmonic Multistage Cyclic Symmetry Analysis.

1.4.5. Automatic Inertia Relief

1.4.6. Terminating and Restarting a Harmonic Balance Method Analysis

1.4.7. Large Mass Method (LMM) Enforced Motion Analysis

1.4.1. Rotordynamics Analysis

A new procedure for the Campbell analysis of a prestressed structure using linear perturbation is now supported ([Alternative procedure for rotational velocity independent prestress in the Rotordynamic Analysis Guide](#)). It is recommended if the prestress is independent from the rotational velocity.

Spring-damper (**COMBIN14**) and bushing (**COMBI250**) elements with tabular real constants (frequency is the primary variable) are now fully supported in a linear perturbation harmonic analysis.

Bearing element ([COMBI214](#)) with tabular real constants (rotational velocity is the primary variable) is now fully supported in a linear perturbation modal and a linear perturbation synchronous harmonic analysis ([SYNCHRO](#) command).

1.4.2. Static Multiframe Restarts for Cyclic Symmetry and Multistage Cyclic Symmetry Analyses

Static multiframe restarts are now supported for [Cyclic Symmetry](#) and [Multistage Cyclic Symmetry Analyses](#). This feature enables you to save information for multiple loadsteps and substeps and restart from a specific loadstep and substep, making analysis corrections as needed.

1.4.3. Node-based Mode File Scoping for Mode-Superposition Analysis

To reduce the .mode file size and improve efficiency in a mode-superposition analysis where displacement solution is of primary interest, writing to the .mode file can be scoped based on a subset of nodes (see [Node-based Mode File Scoping for Mode-Superposition Analysis in the Structural Analysis Guide](#)).

1.4.4. Python Script to Calculate Relevant Harmonic Indices for a Multiharmonic Multistage Cyclic Symmetry Analysis.

Selecting the proper harmonic indices participating in the response is of paramount importance to efficiently solve a multistage cyclic symmetry analysis. A Python script has been developed to facilitate the selection process. Use it to calculate relevant harmonic indices to include for an accurate response in a specified nodal diameter range in a multiharmonic multistage cyclic symmetry analysis. For details, see [Python Script for Selection of Multistage Harmonic Indices in the Multistage Cyclic Symmetry Analysis Guide](#).

1.4.5. Automatic Inertia Relief

An [automatic inertia relief option](#) is available to calculate the residual vectors in a modal analysis, which improves the accuracy for mode superposition analysis (transient and harmonic). For more information, see [Including Inertia Relief Calculations in the Basic Analysis Guide](#).

1.4.6. Terminating and Restarting a Harmonic Balance Method Analysis

There is now a convenient way to terminate an HBM analysis and write the last converged substep to the results (.rst) file. You can then restart the analysis with a new input file, using the last converged result as a solution guess for the next frequency/substep. For more details, see [Aborting an HBM analysis in the Harmonic Balance Method Analysis Guide](#).

1.4.7. Large Mass Method (LMM) Enforced Motion Analysis

Large Mass Method (LMM) enforced motion analysis is now implemented natively. This makes performing LMM analyses easier.

To perform a LMM analysis, first define a large mass by using the [MASS21](#) element. Then, use the [FVAL](#) command to specify displacement, velocity, or acceleration as input. For more information, see the discussion of this command under [New Commands \(p. 68\)](#).

1.5. Transient Dynamics

Release 2025 R2 includes the following enhancements for structural analyses involving transient dynamics:

1.5.1. Adaptive Mass Proportional Damping

1.5.1.1. Adaptive Mass Proportional Damping

You can now use adaptive mass proportional damping to solve complex, non-linear problems. Specify the *AUTO* field in the **ALPHAD** command. Automatically adjusting the mass matrix multiplier for damping improves convergence in non-linear problems that use the quasi-static method (**TINTP,QUASI**) or the static-to-transient method (**SOLOPTION,STOT**).

The following types of problems are expected to benefit from adaptive mass proportional damping:

- Local buckling
- Underconstrained or unconstrained models
- Models where a contact might be lost during the solution

To ensure an accurate solution, the damping is set high enough to provide some stability but low enough that damping energy is a small fraction of the potential energy of the system.

2. Multiphysics

Release 2025 R2 includes the following enhancements for analyses involving multiphysics environments:

2.1. Acoustics

2.2. Thermal

2.3. Coupled-Field

2.1. Acoustics

The following enhancement for acoustic analysis is available:

2.1.1. Irregular Perfectly Matched Layer (IPML) for Axially Symmetric Elements

The 2D acoustic fluid elements, **FLUID243** and **FLUID244**, support the axisymmetric IPML option for harmonic acoustic simulations.

2.2. Thermal

There are no new features and enhancements for thermal analysis this release.

2.3. Coupled-Field

The following enhancements are available for analyses involving coupled-field elements:

2.3.1. Nonlocal Damage Formulation Support

Nonlocal damage formulation is now supported for [SOLID226](#) and [SOLID227](#) for structural-thermal, structural-diffusion and structural-thermal-diffusion-analyses. [SOLID226](#) and [SOLID227](#) are now enabled to activate nonlocal damage models such as generalized damage (implicit gradient regularization). Nonlocal damage formulation capabilities can better capture the distributed nature of damage in real materials by averaging variables over a finite neighborhood and introducing a characteristic length scale. This leads to more objective and reliable results, improved convergence stability and mesh insensitivity in the presence of strain localization or softening.

2.3.2. Coupled-field UPFs: Customizable Element Matrices and Load vectors

User-programmable features (UPFs) available for coupled-field elements [PLANE222](#), [PLANE223](#), [SOLID225](#), [SOLID226](#), and [SOLID227](#) now enable you to customize element matrices and load vectors, thus providing complete control over element behavior. Newly added procedures simplify the access, calculation, and modification of these matrices and vectors. With this enhancement, you can now improve the existing element formulation or completely overwrite it. For nonlinear loads or material models, you can calculate the respective tangent matrices and add them to the element stiffness matrix to improve the convergence of a nonlinear analysis. You can also create custom stiffness, mass, and damping matrices and load vectors to replace the default ones. New examples have been added to the documentation to demonstrate the use of Coupled-field UPFs. For more information, see [Customizing a Coupled-Field Element in the Programmer's Reference](#)

3. Solvers

Release 2025 R2 includes the following improvements to the solution process:

- [3.1. Distributed-Memory Parallel Processing Enhancements](#)
- [3.2. GPU Acceleration Feature Updates](#)
- [3.3. Updated List of Recommended Devices for the GPU Accelerator](#)
- [3.4. Mixed Equation Solver Enhancements](#)

3.1. Distributed-Memory Parallel Processing Enhancements

Enhancements include:

- The default support for message passing interface (MPI) software has been upgraded to Intel MPI 2021.14.2 for both Linux and Windows. Previously, the default was Intel MPI 2021.13.0.

For details and other optional supported MPI software versions, see [MPI Software in the Parallel Processing Guide](#).

3.2. GPU Acceleration Feature Updates

Updates include:

- The ROCm libraries have been updated to 6.3.3. The prior release used version 6.2.0.
- The CUDA libraries have been updated to 12.8. The prior release used version 12.6.

- The performance of the PCG solver has been improved when running with GPU acceleration as more equation solver calculations are offloaded on the GPU card(s).

For more information on GPU cards and their updates, see [Requirements for the GPU Accelerator in Mechanical APDL \(Linux\)](#) and [Requirements for the GPU Accelerator in Mechanical APDL in the Ansys, Inc. Installation Guides](#) (Windows).

3.3. Updated List of Recommended Devices for the GPU Accelerator

For Linux and Windows, the following cards have been tested and added to the list of recommended GPU devices for the GPU accelerator:

- NVIDIA B300
- NVIDIA B200
- NVIDIA H200
- NVIDIA RTX Pro 6000
- NVIDIA RTX Pro 5000

For the complete list of recommended GPU accelerators, see [Requirements for the GPU Accelerator in Mechanical APDL \(Linux\)](#) and [Requirements for the GPU Accelerator in Mechanical APDL in the Ansys, Inc. Installation Guides](#) (Windows).

3.4. Mixed Equation Solver Enhancements

The mixed equation solver now supports all static thermal analyses except for quasi thermal analyses (**THOPT**,QUASI).

4. Other Enhancements

The following other enhancements have been added:

- 4.1. CAD Support Updates
- 4.2. Element Face Components to Represent Surfaces
- 4.3. New Coded Database File Command
- 4.4. Modified Coded Database File Commands
- 4.5. Element Single-Value Results Supported for SHELL281
- 4.6. Parameter Name Limit Increased to 240 Characters within Mechanical APDL
- 4.7. Component Names Support Special Characters

4.1. CAD Support Updates

The following CAD support updates are available for the Mechanical APDL software:

Added support for –

- NX 2506 (limited to Windows 10 and 11 Professional and Enterprise versions)

- NX 2412 (limited to Windows 10 and 11 Professional and Enterprise versions)
- Parasolid 36.1 (Windows 10, Windows 11, and Linux)

Removed support for –

- NX 2306 (on all operating systems)
- Parasolid 35.1 (Windows 10, Windows 11, and Linux)

For more information, see [Introduction to the Connection Functionality in the *Connection User's Guide*](#).

4.2. Element Face Components to Represent Surfaces

Element face (EFACE) components are now available in the Mechanical APDL product. These components represent surfaces using their combination of element IDs and corresponding aggregated bitmap face ID values. EFACE components use a compact format by default which significantly reduces memory usage. For more information, see [CMBLOCK Command in the *Programmer's Reference*](#), **CMLIST**, and ***VGET**.

4.3. New Coded Database File Command

The following new coded database (CDB) file commands are available:

- **/BLOCK** – Notifies the program when block input starts and ends. This command may improve performance by enabling more efficient reading of the specified data.

4.4. Modified Coded Database File Commands

The following CDB file commands have been enhanced or otherwise modified:

- **EBLOCK** – Defines a block of elements. **EBLOCK** now supports defining components.
- **NBLOCK** – Defines a block of nodes. **NBLOCK** now supports defining components.

4.5. Element Single-Value Results Supported for SHELL281

The **SHELL281** element type now supports single-value results. For more information, see [Solution Output in the *Element Reference*](#).

4.6. Parameter Name Limit Increased to 240 Characters within Mechanical APDL

The maximum number of characters for all parameter names in Mechanical APDL has been raised to 240 characters. In the previous release, the maximum of 240 characters for parameter names was beta, with the exception of long tables.

For more information, see ***SET**, **SUGET**, and ***DIM**.

4.7. Component Names Support Special Characters

Component names now support a variety of special characters. For more information, see **CM** and **CMGRP**.

5. Commands

This section describes changes to commands at Release 2025 R2:

5.1. New Commands

5.2. Modified Commands

5.3. Undocumented Commands

Some commands are inaccessible from menus and are available via the **command-input field** or batch-file input only. The documentation for each command indicates menu path information, if available.

5.1. New Commands

The following commands are new for this release.

- **CNTGEN** – Generates a contact pair based on components. The command creates a contact pair based on element components with face ID (EFACE component(s) via the **CMBLOCK** command) or nodal components provided for the contact surface and the target surface.
- **FVAL** – Defines a Large Mass (LM) force for base excitation problems. The command takes displacement, velocity, or acceleration as the input and converts it to a force that is applied in the specified direction. You must add a **MASS21** element at the specified node to compute the LM force.
- **SFCGEN** – Generates multi-point (MPC) surface-based constraints based on element components. The command creates surface-based MPCs with face ID (EFACE component(s) via the **CMBLOCK** command) or nodal components provided for the contact surface and the target surface.

5.2. Modified Commands

The following commands have been enhanced or otherwise modified:

- **ALPHAD** – Enables adaptive mass proportional damping via the *AUTO* option. This improves convergence in non-linear problems that use the quasi-static method or the static-to-transient method.
- **AMBUILD** – Specifies printer parameters for the build and other options in an additive manufacturing (AM) analysis. A new **AMBUILD,SEQUENCE** option allows you to define distinct heating steps in an electron beam melting (EBM) process as well as component-specific process parameters for both laser powder bed fusion (LPBF) and EBM.
- **CM** – Groups geometry items into a component. Component names now support a set of special characters.
- **CMGRP** – Groups components and assemblies into an assembly. Component names now support a set of special characters.

- **CMLIST** – Lists the contents of a component or assembly. A new EFACE entity type enables listing element face components.
- **/CONFIG** – Assigns values to Mechanical APDL configuration parameters. The default for **/CONFIG,RESUPREC** is now $VAL = 1$ if **/CONFIG** is not issued or **/CONFIG,RESUPREC** is issued without a value.
- **DELTIM** – Bypasses the default automatic time incrementation via the *ForceKey* option, The program adheres to the initial time increment specified.
- ***DIM** – Defines an array parameter and its dimensions. The parameter name (*Par*) can now contain up to 240 characters.
- **ESOL** – Specifies element data to be stored from the results file. The existing *CAP* item has been extended to the EDP Cap Creep model. Use the new *CAPC* item to post-process, in the EDP Cap Creep model, the equivalent creep strains used in the compaction zone and the shear and expansion zone.
- **ETABLE** – Fills a table of element values for further processing. The existing *CAP* item has been extended to the EDP Cap Creep model. Use the new *CAPC* item to post-process, in the EDP Cap Creep model, the equivalent creep strains used in the compaction zone and the shear and expansion zone.
- ***GET** – Retrieves a value and stores it as a scalar parameter or part of an array parameter. Use the new postprocessing data type output ($IT3NUM = DKTH$ when Entity = CINT) to retrieve the threshold stress-intensity factor range specified in SMART fatigue crack-growth analysis.
- **MODCONT** – Specifies additional modal analysis options. Use the new *FastLV* option, NOEXP, to activate fast load vector generation without modal expansion and reduce writing to the .mode file by activating [node-based mode file scoping](#).
- **OSRES** – Controls the selected result data written to the database, helping to reduce the size of the results (RST) file. The initial state values for various quantities can now be stored and postprocessed using the **INISTATE** command.
- **PLCINT / PRCINT** – Plots / lists fracture parameter (**CINT**) results data. Use the new data type output ($Dtype = DKTH$) to retrieve the threshold stress-intensity factor range specified in SMART fatigue crack-growth analysis.
- **PLESOL** – Displays solution results as discontinuous element contours. The existing *CAP* item has been extended to the EDP Cap Creep model. Use the new *CAPC* item to post-process, in the EDP Cap Creep model, the equivalent creep strains used in the compaction zone and the shear and expansion zone.
- **PLNSOL** – Displays solution results as continuous element contours. The existing *CAP* item has been extended to the EDP Cap Creep model. Use the new *CAPC* item to post-process, in the EDP Cap Creep model, the equivalent creep strains used in the compaction zone and the shear and expansion zone.
- **PRESOL** – Prints the solution results for elements. The existing *CAP* item has been extended to the EDP Cap Creep model. Use the new *CAPC* item to post-process, in the EDP Cap Creep model, the equivalent creep strains used in the compaction zone and the shear and expansion zone.

- **PRNSOL** – Prints nodal solution results. The existing *CAP* item has been extended to the EDP Cap Creep model. Use the new *CAPC* item to post-process, in the EDP Cap Creep model, the equivalent creep strains used in the compaction zone and the shear and expansion zone.
- **RBGEN** – Via the *ECOMP* option, optionally specifies the name of a new component that is composed of new rigid elements generated by the command.
- **RESWRITE** – Appends results data from the database to a results file. Use the new *gFlag* option to append results data from the database to the results file, if there is no geometry data in the file.
- **RMODIF** – Optionally, specify an acronym name instead of a serial number when you are using this command to modify a real constant item for contact elements.
- ***SET** – Assigns values to user-named parameters. The parameter name (*Par*) can now contain up to 240 characters.
- **SUGET** – Moves surface geometry and mapped results to an array parameter. The parameter name (*Parm*) can now contain up to 240 characters.
- **TB** – Activates a data table for linear and nonlinear material properties or special element input.
 - A new [frictional contact model](#) is now available using *FRIC, TBOPT=ATAN*.
 - Anisotropic electrical conductivity is now supported with a new material model data type (**TB,ELEC**).
- ***VGET** – Retrieves values and stores them into an array parameter. A new *EFACE* keyword for the *Entity* option returns aggregated bitmap face ID values of corresponding elements on surfaces.

5.3. Undocumented Commands

No commands have been undocumented this release.

For information about commands that have been undocumented in prior releases, see the [archived release notes](#) for Mechanical APDL.

6. Elements

This section describes changes to elements at Release 2025 R2:

[6.1. New Elements](#)

[6.2. Modified Elements](#)

[6.3. Undocumented Elements](#)

Some elements are not available from within the GUI. For a list of those elements, see [GUI-Inaccessible Elements](#).

6.1. New Elements

The following new elements are available at this release:

- **MPC184-Genc**- A two-node joint element in which the relative rotations are characterized by the rotation axis that positions the local coordinate system at one node with respect to the local coordinate system at the other node. The element is used to connect two bodies to transmit motion from one body to another. Stiffness and damping behavior can be specified for this joint element.
- **SHELL229**- A 3D layered shell element with in-plane thermal and electrical capabilities (membrane behavior). It has no limitation on the number of material layers. The element has four nodes and supports two degrees of freedom per each node, temperature and voltage. It supports the Joule heating effect and generates temperatures that can be passed to structural shell elements in order to model thermal strain. For verification examples, see [VM215](#).

6.2. Modified Elements

The following elements have been enhanced:

- **CONTA172** and **CONTA174** - The option KEYOPT(12) = 2 changed to no separation (penetration only).
- **CONTA175** and **CONTA177** - The following changes have been made to these elements:
 - The double-sided target surface option (KEYOPT(8) = 1) is now valid for **CONTA175**.
 - The mixed constraint type option (KEYOPT(5) = 2 of target element) is extended to these two elements.
 - The option KEYOPT(12) = 2 changed to no separation (penetration only).
- **MPC184** - This element has been modified for instances when a joint element node is not connected to the rest of the body (free-hanging node), it is not a grounded node, and no boundary conditions are imposed on such a node. In these cases, the joint element is treated as a weld joint.
- **MPC184-Spotweld** - This element has been modified for cases when it is used with a solid element. In these instances, the rotational degrees of freedom at the node(s) of the spotweld joint are automatically constrained.
- **SOLID186**, **SOLID187**, **SOLID226**, and **SOLID227** - These elements are now enabled to activate non-local damage models such as generalized damage (implicit gradient regularization).
- **SOLID272** and **SOLID273** - The **MODE** command is now supported using KEYOPT(2) = 1. This allows the use of the master plane for non-axisymmetric loading. KEYOPT(3) has been added and when used with KEYOPT(2) = 1, all nodal results are saved to the .rst file. This enables 3D plotting with pseudo-nodal planes set by the **NAXIS** command. Each mode shape can be visualized and combined with various mode shapes during post-processing.
- **SOLID272** - KEYOPT(1) has been added to support the enhanced strain formulation.

6.3. Undocumented Elements

No elements have been undocumented this release.

For information about elements that have been undocumented in prior releases, see the [archived release notes](#) for Mechanical APDL.

7. Documentation

Ansys continues to refine the Mechanical APDL documentation set. To that end, the following notable changes and enhancements to the documentation have occurred:

7.1. Updates for Programmers

Routines and functions documented in the [Programmer's Reference](#) have been updated to reflect the current source code. To see specific changes in a file, Ansys recommends opening both the old and current files (using a text editor that displays line numbers), then comparing the two to determine which lines have changed. You can copy the updated files to your system by performing a custom installation of the product.

7.2. Feature Archive

Legacy features, commands, elements, and theory information continue to move to the [Feature Archive](#). While Ansys intends to support legacy capabilities for the immediate future, some may be undocumented in future releases. Consider moving to their recommended replacements. For information about especially notable features that have been archived, see [Features Removed or Archived \(p. 75\)](#) in the Update Guide section of this document.

8. Technology Showcase: Example Problems

The analyses presented in the [Technology Showcase: Example Problems](#) demonstrate the extraordinarily broad simulation capabilities of Ansys Mechanical APDL. The real-world problems highlight the features and effectiveness of Mechanical APDL by presenting a series of analyses from a variety of engineering disciplines.

The problems are more substantive and complex than examples found in the standard documentation set. The documentation thoroughly examines the physics involved with each problem and the considerations necessary for translating problems into numerical models. Approximation issues, accuracy considerations, and recommended practices are discussed. For more information, see [What You Need to Know](#).

The following example problems have been added at this release:

- TD-72 - [Two-Rotor Compressor Analysis Using Multistage Cyclic Symmetry](#)

Many of the Mechanical APDL Technology Showcase problems can also be performed entirely within the Mechanical Application. For more information, see the [Mechanical Technology Showcase: Example Problems in the Mechanical Technology Showcase: Example Problems](#).

9. Mechanical APDL Release 2025 R2 Update Guide

This section contains information about feature enhancements that can affect program behavior or analysis results in ways that you may not expect. Also covered are known incompatibilities, notable issues and defects that have been resolved, and information about replacement capabilities for features that have been removed.

The following topics offer supplemental 2025 R2 product-update information presented by the Mechanical APDL development and testing teams:

- 9.1. Backward Compatibility
- 9.2. Feature Updates Causing Result or Behavior Changes
- 9.3. Known Incompatibilities
- 9.4. Known Issues
- 9.5. Resolved Issues and Defects
- 9.6. Features Removed or Archived

If you are upgrading across several releases, you may find it helpful to consult the Update Guide sections of the [archived release notes](#) for Mechanical APDL.

For information about past, present, and future operating system support, see the [Platform Support](#) section of the [Ansys Website](#).

9.1. Backward Compatibility

Mechanical APDL Release 2025 R2 can read database files from all prior Mechanical APDL releases. Due to ongoing product improvements and defect resolutions, however, results obtained from old databases running in new releases may differ somewhat from those obtained previously.

9.2. Feature Updates Causing Result or Behavior Changes

The following Mechanical APDL feature updates in Release 2025 R2 are known to produce program behaviors or analysis results that differ from those of the prior release:

- 9.2.1. Change in Harmonic Analysis with CORIOLIS,ON
- 9.2.2. Change in Enforced Motion Calculation
- 9.2.3. Change in Automatic Inertia Relief Behavior
- 9.2.4. Change in MNF File for Special and Service Releases
- 9.2.5. *DMAT: Matrix Type Incompatibility No Longer Accepted for MMF file

9.2.1. Change in Harmonic Analysis with CORIOLIS,ON

In a full harmonic analysis with Coriolis effect activated in the rotating reference frame (**CORIOLIS,ON**), the centrifugal force is not taken into account. In past releases, it was considered a harmonic load.

9.2.2. Change in Enforced Motion Calculation

If degree of freedom coupling or constraint equations are present, the [enforced motion calculation](#) now provides more robust solutions. For large models, it may lead to longer computation time.

9.2.3. Change in Automatic Inertia Relief Behavior

If the rigid body modes generated using the geometry ($RIGID_CALC = 1$) are found to be insufficiently accurate, the program calculates the rigid body modes again using an eigensolver.

9.2.4. Change in MNF File for Special and Service Releases

In Special Version and Service Package releases, the ADAMS reader now automatically truncates the release name in MNF files to ensure they do not exceed the 16-character limit. This ensures compatibility and prevents crashes during file processing.

9.2.5. *DMAT: Matrix Type Incompatibility No Longer Accepted for MMF file

When importing a dense matrix from an MMF file, you must ensure that the specification of *Type* on ***DMAT** matches the MMF file's declared data type. Previously, mismatched types were automatically converted without error. Starting at release 25.2, incompatible types trigger an error: *"*DMA Command : The involved objects have inconsistent matrix types."*

9.3. Known Incompatibilities

The following incompatibilities with prior releases are known to exist at Release 2025 R2:

- **Change in force summation when post-processing a Dynamic Design Analysis Method (DDAM) spectrum analysis** - Starting at 2025 R2, the spectrum analysis summation is used when issuing the commands **PRRFOR**, **PRNLD**, **NFORCE**, and **FSUM**. The forces are summed before they are combined.

9.4. Known Issues

The following issues are known to exist at Release 2025 R2:

- **Linear Perturbation analysis limitation** – The total tangent stiffness matrix obtained from a Linear Perturbation basis analysis may include stress stiffness and load stiffness contributions (geometric stiffness). If the geometric stiffness contribution is significant, then the total tangent stiffness matrix, when used in a Linear Perturbation downstream analysis, may produce reaction forces not satisfying the equilibrium with the external forces.
- **GPU acceleration hangs on AMD Cards with small models** – When running certain small models with the GPU accelerator enabled on AMD cards, specifically using three or more processes, the program may run very slowly and appear to hang.

Workaround: To avoid this issue, create the following environment variable:

GPU_MAX_HW_QUEUES=2

- **Automatic initialization in curve fitting requirement** – To use automatic initialization in curve fitting for hyperelastic models with Prony series viscoelasticity, the latest [AML Python module](#) must be installed beforehand.

For issues discovered following publication of this document, see [Mechanical APDL in the Ansys Known Issues and Limitations](#).

9.5. Resolved Issues and Defects

The following notable issues and defects for Mechanical APDL have been resolved at Release 2025 R2:

ID	Resolution Description
1174494	An instability when using a *PYTHON block containing Mechanical APDL errors in batch mode has been resolved.
1247948	The instability when running <code>ans_admin.exe</code> without an Intel compiler has been resolved.
1172441	The problem with left-clicking external links in the documentation, which previously did not open the intended website, has been fixed. Left-clicking an external link now correctly opens the website in a new tab.
1159211	The limitation regarding nesting while using Open MPI on AMD GPU devices has been documented. For more information, see GPU Accelerator Capability in the Parallel Processing Guide .
1139164	The issue with using the PCG solver on AMD GPU devices has been resolved.
1164591	The ADAMS reader previously crashed when processing MNF files containing release names that were longer than 16 characters in Special Version (SV) and Service Package (SP) releases. This issue has been resolved. For SV and SP releases, the release names in MNF files are now automatically truncated in the output. For more details, see Change in MNF File for Special and Service Releases .

9.6. Features Removed or Archived

If a legacy Mechanical APDL feature has been removed at Release 2025 R2, this section provides information about the Mechanical APDL replacement feature, its functional equivalent in another Ansys product, or another workaround.

Legacy features that have been [archived](#) also appear here. While archived features remain available for use, technical enhancement is unlikely to occur, and better alternatives are available and recommended in most cases.

The following topics are available:

[9.6.1. MSAVE](#)

[9.6.2. KEYOPT 18 for CPT elements](#)

9.6.1. MSAVE

The **MSAVE** command has been undocumented. Advancements to the Preconditioned Conjugate Gradient (PCG) solver and **PCGOPT** command make **MSAVE** obsolete.

9.6.2. KEYOPT 18 for CPT elements

KEYOPT 18 has been undocumented for the CPT elements. When defining the nonlocal damage material models (generalized damage, microplane regularized damage, such as elastic and coupled damage-plasticity, and regularized anisotropic damage), you no longer need to define KEYOPT 18 for CPT elements. Previously, KEYOPT 18 had to be set to certain model-specific values to activate the required extra degree(s) of freedom. Now, the extra DOF(s) are automatically activated with a non-zero length scale parameter definition.

Autodyn Release Notes

The Ansys Autodyn product encompasses all of the following explicit solvers: FE (Lagrange), Euler, FCT, ALE, and SPH, and various means to couple them together. All are integrated into the Autodyn Component system, while the FE (Lagrange) and Euler—including Euler-Lagrange coupling—are also integrated into the Explicit Dynamics Analysis system.

1. New Features and Enhancements

1. New Features and Enhancements

The following enhancements were made for Release 2025 R2:

- **Intel MPI upgrade.** The default version of Intel MPI used to run parallel Autodyn jobs has been upgraded to version 2021.14.2.

Aqwa Release Notes

This release of the Aqwa related products contains all capabilities from previous releases. The following enhancements are available in release 2025 R2. Refer to the product-specific documentation for full details of the new features.

1. [Aqwa Solver Modules](#)
2. [Aqwa Graphical Supervisor \(AGS\)](#)
3. [Hydrodynamic Analysis Systems](#)

1. Aqwa Solver Modules

The following new features provide extended capabilities in the Aqwa solver modules:

- **Deactivating Degrees of Freedom.** The new FDOF data record is used to define the deactivated degrees of freedom of a specified structure. The reaction force or moment at the structure's COG is calculated and output. Ansys recommends using this data record to replace the DACF data record.
- **Recommended timestep for stability.** Considerations for setting the time step interval are described in the documentation for the TIME data record. A runtime warning message is issued if the time step interval is greater than the minimum value determined by the wave frequency and the analysis type.
- **Moonpool analysis in Aqwa Librium and Aqwa Fer.** The coupling effects of fluid motions inside a moonpool and outside the external hull, and the rigid body motions of structures, are estimated in Aqwa Librium and Aqwa Fer analyses. The eigenvalues and frequency statistical results include the moonpool free surface pressure mode response components.

2. Aqwa Graphical Supervisor (AGS)

The following new feature provides extended capabilities in the Aqwa Graphical Supervisor (AGS):

- **Eigenvalues in the online Dynamic Stability analysis.** The listed eigenvalues include the moonpool free surface pressure mode response components in the AGS Dynamic Stability analysis.

3. Hydrodynamic Analysis Systems

The following new features provide extended capabilities in the Hydrodynamic systems in Workbench:

- **Linear cable stiffness.** You can define the stiffness of a linear elastic cable by the Elastic Modulus, EA.

- **MCOL Hydrodynamic Database output file option.** You can output the hydrodynamic database in a text-based .mco format, which can then be read by the LS-DYNA MCOL solver for further ship grounding and collision event simulation.
- **Watch Zones.** You can add and define Watch Zones for various marine operations. The trajectory of the tracked points with the selected watch zones is displayed in the Stability and Time response analyses. The overshooting statistic results are given for the Hydrodynamic Response Time Domain analysis.
- **Smaller mass per length.** The minimum limit for **Mass / Unit Length** in a **Catenary Section** is defined as 0.02% of the defined water density (approximately 0.2 kg/m).
- **Multi-selection editing.** If you select multiple objects of the same type in the Outline tree and make changes in the Details pane, the changes are applied to all selected objects.
- **Fixed Points in Wave Surface Elevation graph.** The Reference Point input, to select the position at which the Wave Surface Elevation is reported, can be a Fixed Point.
- **Display properties of selected geometry and mesh entities.** You can select one or more geometry vertices, edges, faces, or bodies; or mesh nodes or elements in the Graphical window to display their properties.
- **Geometric symmetry.** Half or quarter parts of the symmetric structure geometry (including Surface Bodies, Line Bodies, Discs and External Lids) can be meshed and used for increasing the accuracy and efficiency of the Hydrodynamic Diffraction analysis.

Ansys Composite Prep (ACP) Release Notes

This release of the Ansys Composite Prep™ (ACP) layered composite modeling and preprocessing software contains all capabilities from previous releases. The following sections present the new features, enhancements, and known limitations for the ACP application in Release 2025 R1.

- 1. New Features and Enhancements
- 2. Known Limitations

1. New Features and Enhancements

The ACP application includes new features and enhancements at Release 2025 R1 in the following areas:

- 1.1. Transformation Wizard for Rosette Definitions
- 1.2. Symmetric Layups: Copy and Paste in Reverse Order
- 1.3. Automatic Naming of Modeling Plies
- 1.4. 3D Solid Models: Ply Thicknesses at Nodal Locations for CAD Geometries
- 1.5. Lookup Tables: Definition of Variable Data
- 1.6. PyACP Enhancements
- 1.7. Postprocessing through PyDPF Composites
- 1.8. Ansys Composite Cure Simulation (ACCS) Updates

1.1. Transformation Wizard for Rosette Definitions

A transformation wizard is available to simplify the definition of rosettes. You can enter and execute translations and rotations individually, increasing the accuracy of the layup model.

For more information, see [Rosettes in the ACP User's Guide](#).

1.2. Symmetric Layups: Copy and Paste in Reverse Order

Layups in composite structures usually require symmetry from a structural or regulation point of view. The new **Copy and Paste in Reverse Order** feature simplifies the definition of symmetric laminates.

For more information, see [Modeling Ply Properties in the ACP User's Guide](#).

1.3. Automatic Naming of Modeling Plies

You can automatically set modeling ply names based on a user-defined format. For example:

{group name} - {material name} - {ply angle}

The automatic naming of modeling plies enables identifying plies while optimizing laminates on a failure analysis or general postprocessing. The feature appears as **Rename Based on Definition** and **Append Name Based on Definition** in the context menu of the Modeling Groups folder, Modeling Group, and Modeling Plies.

For more information, see [Modeling Groups in the ACP User's Guide](#).

1.4.3D Solid Models: Ply Thicknesses at Nodal Locations for CAD Geometries

You can now compute ply thicknesses at nodal locations for CAD geometries, enabling a more accurate 3d layered model extrusion. The feature already supports [cut-off selection rules](#), [lookup tables](#), and [ply tapering](#). For more information on the feature for geometries, see [Thickness in the ACP User's Guide](#).

1.5. Lookup Tables: Definition of Variable Data

[Lookup tables \(LUT\)](#) now enable the definition of variable data such as [thickness distribution](#), [fiber orientation](#), [shear angle](#), and [field definition](#). You can define the [3D LUT](#) locations and directions with the [rosette's](#) custom coordinate system. In prior releases, the inputs were only based on the global coordinate system.

The definition of variable data feature simplifies the input in models or assemblies that have changed locations due to new designs. The feature enables using a rosette that moves with the respective coordinates and defines the inputs relative to that coordinate system.

1.6. PyACP Enhancements

, the public Python module containing ACP preprocessing functionality, has several enhancements that can be used to define layups for complex structures. PyACP enhancements enable you to do the following:

- Scope [sensors](#) to a [solid model](#).
- List analysis plies of a [layup mapping object](#) or [imported solid model](#).
- List [interface layers](#) of a [solid model](#).
- Create models with the beta to [PyMechanical](#) workflow that can be postprocessed in [PyMechanical](#) or the [Mechanical](#) product's standalone application.

If you have any questions or feature requests, contact the composites team at composites@ansys.com or create an issue on [GitHub](#).

1.7. Postprocessing through PyDPF Composites

The Python module contains the following enhancements for postprocessing composite structures:

- The sampling point feature is available for use with solid models, enabling the extraction and visualization of the through-the-thickness distribution of results. The feature automatically collects all solid elements of the laminate stack and provides the results from the bottom to top layer of the laminate. For more information, see the example in [Example 1.7.1](#).

- PyDPF Composites can postprocess results from the LS-DYNA product. The Python module can filter data and customize analyses. For more information on setting up the LS-DYNA product's simulation and postprocessing with the Python module, see the example in .

1.8. Ansys Composite Cure Simulation (ACCS) Updates

Updates include:

- The performance of the RTM solver has been improved.
- The postprocessing efficiency has been improved through result extraction enhancements.
- ACCS now supports transferring degrees of cure from RTM simulations to cure simulations. You can pass the changing degree of cure from the infusion process to subsequent analyses to simulate the curing remainder.

For more information, see [Ansys Composite Cure Simulation Documentation](#).

2. Known Limitations

General ACP application and release-specific limitations can be found in [Known Limitations in the ACP User's Guide](#).

Material Designer Release Notes

This release of the Ansys Material Designer™ material homogenization software contains all capabilities from the previous release. The following sections present the notes for the Ansys Material Designer application's 2025 R2 release.

1. Notes

Material Designer Installation

To install the Material Designer application, you must select the Material Designer checkbox in the installation. The Material Designer product will no longer automatically be installed with the **Mechanical Products**.

Alternatively, you can use the `-matdesigner` flag in the silent installer. For more information, refer to the [Ansys, Inc. Installation Guides](#).

2. Known Limitations

General Material Designer application and release-specific limitations can be found in [Known Limitations in the Material Designer User's Guide](#).

Additive Manufacturing

Additive Suite products include new features and enhancements at Release 2025 R2 in the following areas:

1. LPBF Process Simulations
2. DED Process Simulations
3. Distortion Compensation
4. Changes in Product Behavior
5. AM-Related Mechanical APDL Commands

1. LPBF Process Simulations

Enhancements for powder bed fusion (PBF) simulations in the Ansys Mechanical™ application include the following:

- **Quick Calibration:** A traditional full-simulation calibration is time-consuming, requiring multiple simulation iterations to meet a set tolerance target. In contrast, Ansys's new [Quick Calibration feature](#) uses a machine-learning approach on a common cantilever geometry with support cutoff to significantly reduce LPBF calibration time from hours to minutes, or even seconds, while maintaining high accuracy. This time-saving benefit is especially valuable for complex, location-specific strain scenarios that require multiple pairs of distortion targets.
- **Location specific strain scaling factors:** You can now specify [strain scaling factor \(SSF\) values that vary in space](#) to account for variability in strains for different locations on the build plate.
- **Directional recoater interference:** You can now capture the effects of recoater direction in the [recoater interference result](#) for LPBF analyses. The processing for this specific capability is done in post processing, where you can define different recoater directions and view how those different directions affect the results without re-solving the analysis.
- **Machine Learning Prediction:** The ML model (for [Thermal Strain Method](#) = Machine Learning Prediction) has been retrained using a new underlying algorithm based on cooling cycles rather than the thermal ratcheting algorithm. This new algorithm is more accurate in following experimental distortion trends as machine parameters are varied. Ansys recommends you perform a Strain Scaling Factor calibration specifically for this new ML Thermal Strain method.
- **UI tooltips for AM objects:** Hover over the options in the Details pane to see helpful tooltips for the AM-specific objects of [AM Bond](#), [AM Process](#), [Build Settings](#), [Support Group](#), [Generated Support](#), [Predefined Support](#), and [STL Support](#), as well as the AM options in Analysis Settings.
- **LPBF Shrinkline result (Beta):** A new shrinkline result allows you to pinpoint areas where one or more layers are contracting more than the surrounding material—this localized contraction is commonly referred to as a shrinkline.

- **Visualize knockdown factors for voxel mesh (Beta):** You can now visualize knockdown factors when using the [Voxelizer option in the Cartesian mesh method](#). The option is available under Display Type > View on the AM Process object when the Cartesian Mesh Method has the `Mesh Using Voxelization` set to True or if an STL Support is present in the Tree. This is similar to the capability that is already in place for visualizing knockdown factors for [STL Supports](#).

2. DED Process Simulations

Enhancements for directed energy deposition (DED) simulations using the [DED Process Add-on](#) include the following:

- **Path to Part modeling:** You can now [generate geometry models directly from G-code](#) for your simulations. This method offers more accurate views of print characteristics, like deposition volume and geometric feature details, compared to using nominal CAD designs. The new Additive DED add-in in the Discovery™ application creates G-code based models quickly with just a few clicks and transfers them seamlessly to the Mechanical application for simulation.
- **Edit cooling and other parameters using cluster output file:** At the last release, we introduced the ability to directly edit certain build parameters by [editing the cluster output file](#), `DEDC\clusterOutput.json`. In the same manner, you can now edit the process temperature or power, room temperature, and the cooling boundary condition parameters of convection coefficient and radiation emissivity for both the part and the base.
- **Improved G-code path visualization for multi-axis models:** The [path visualization options](#) of Path Layers to Show, Show Every *n*-th Layer, and Animation Frames are now available for multi-axis models, where one "layer" is defined by laser status changes. The lines between each pair of laser on and off commands are grouped into a layer.

3. Distortion Compensation

The [Distortion Compensation Add-on](#) is a tool for correcting distortions caused by the manufacturing process. Enhancements include:

- **Export compensated CAD geometry:** You can now [export a compensated geometry](#) in a Discovery CAD (.dsc0) file format. Previously, only an .stl format was available. This feature allows you to transform/morph the original CAD geometry using the deformed compensation data. Internally, this feature uses a Discovery API (the Ansys Discovery™ application must be installed) to perform geometry transformation. At least two successful iterations on an [Iterative Compensation](#) or a [Single Compensation](#) result object are required to generate modified CAD geometry.

4. Changes in Product Behavior

Release 2025 R2 of the Mechanical application includes AM-related enhancements that cause product behavior to differ from earlier releases, as explained below:

Laser Powder Bed Fusion (LPBF)

- The Direct solver is now used instead of the Iterative solver when Solver Type is set to the "Program Controlled" option in Analysis Settings. This change typically improves performance in additive manufacturing (AM) simulations.
- **Machine Learning Prediction:** A new underlying algorithm based on cooling cycles has replaced the thermal ratcheting algorithm for the [Thermal Strain Method](#) of [Machine Learning Prediction](#). This new algorithm is more accurate in following experimental distortion trends as machine parameters are varied. Ansys recommends you perform a Strain Scaling Factor calibration specifically for this new result.

Additionally, the formulation for applying strains on supports has been updated to accommodate the updated Machine Learning Prediction algorithm.

5. AM-Related Mechanical APDL Commands

Changes to the AM-related Mechanical APDL commands include the following:

- **AMBUILD** – Specifies printer parameters for the build and other options in an additive manufacturing (AM) analysis. A new **AMBUILD,SEQUENCE** option allows you to define distinct heating steps in an electron beam melting (EBM) process, as well as component-specific process parameters for both laser powder bed fusion (LPBF) and EBM.

Sherlock

Introduction

This document provides a summary of all new features, product enhancements, limitations, and bug fixes in the 2025 R2 release of the Ansys Sherlock™ electronics reliability prediction software.

1. [Known Issues](#)
2. [Scheduled Updates](#)
3. [General Enhancements and Improvements](#)
4. [API Updates](#)
5. [Bug Fixes](#)
6. [Third Party Dependency Changes](#)

1. Known Issues

When exporting a blocked CDB model from the Sherlock application with solder balls modeled as shells and beams, the solder balls are missing from the exported CDB file. If the **Display Model After Export** option was enabled in the **Export Model** dialog, the application fails to set up the Workbench project and displays an error message. If **Display Model After Export** is *not* enabled, no error is reported unless you try to use the exported file in any application. (1282848)

The Sherlock 2025 R2 Service Pack 1 corrects the above issue. (1292160)

2. Scheduled Updates

Overall Board Score will be removed in the next release of Sherlock, 2026 R1.

3. General Enhancements and Improvements

Links:

- 3.1. [Solder Fatigue Model Updates](#)
- 3.2. [Performance Improvements](#)
- 3.3. [General Enhancements](#)
- 3.4. [Other Improvements](#)

3.1. Solder Fatigue Model Updates

- BGA Solder Fatigue Model:

- The BGA Solder Fatigue model calculation has been updated to use Distance-To-Neutral-Point (DNP) for under-the-die shadow and corner when computing stress and strain. Because of this new DNP functionality, the Solder Fatigue analysis dialog will display an additional warning message if **Ball Modeling** is not enabled for any BGA or CGA part in the Parts List.
- The BGA Solder Fatigue model now uses a BGA Calibration Factor which is configurable for each solder material. The **BGA Calibration Factor** field is located in the **Low-Cycle Fatigue** tab of the **Solder Editor**.
- CGA Solder Fatigue Model:
 - Modified the CGA Solder Fatigue Model to use Distance-to-Neutral-Point (DNP) instead of the half-diagonal length and effective column height (which subtracts the heights of the fillets for the pad and package from the column height) when calculating sheer stress for hot and cold and displacement for hot and cold thermal mismatch. The Solder Fatigue Model also uses a calibration factor that is configurable for each solder material, in the Low-Cycle Fatigue tab of the Solder Editor, the CGA Calibration Factor field.
 - Also, CGA stress and strain calculations were updated to account for pad size effects on column height. The update also includes accounting for the actual column location as opposed to using the half-diagonal length of the package. Also, the Solder Fatigue Model now uses a Calibration Factor that is configurable for each solder material. This **CGA Calibration Factor** field is located in the **Low-Cycle Fatigue** tab of the Solder Editor.
 - CGA related guess values, error validation logic, and messages have been updated.
- IMS Solder Fatigue Model:
 - The **Solder** tab in the **Part Properties Viewer/Editor** has a new field: **Design Factor**. It is used in the Solder Fatigue Analysis for BTC packages. You can use this setting to scale down the life prediction from routed pad to other pad styles. Sherlock uses 30% as the default for plane configuration.
 - You can now use a user-defined temperature dependent modulus for dielectric materials.
 - The analysis now splits the strain energy of a given temperature range (T_{max}, T_{min}) into hot (T_{max}, T_{mean}) and cold (T_{mean}, T_{min}) strain energies.

3.2. Performance Improvements

- Sherlock's FEA PCB modeling performance has improved for Layered and Uniform elements.
- Improved performance of the Python script, which is executed in Ansys Mechanical, that creates Named Selections and assigns hole properties in the Trace Reinforcement Model export .
- Improved performance when loading 3D result files into the Sherlock 3D Viewer.
- Improved Sherlock performance when generating FEA result layers such as displacement and strain plots.

3.3. General Enhancements

- Midpoint nodes for second order element bonds are now included as part of the constraint equation.
- Added a beta feature when enables contact elements in place of constraint equations for FEA models.
- Added an FEA setting **Create Sherlock Model**. When enabled, Sherlock generates a preliminary FEA model file during the FEA analysis which you can view even if the analysis fails. When disabled, model generation time is shortened, but no model is available for viewing if the analysis fails.
- Added an FEA setting **Collect Additional Strain Data**. When enabled, the Sherlock application collects maximum principal strain and directional strains for the ICT analysis which are displayed in the Results Table and Layer Results.
- For CGA components, part property fields no longer use BGA (ball) terminology. For example, **Ball Pitch** is now **Column Pitch** for CGAs.
- You can now right-click a Thermal Map file on the Sherlock **Project Tree** and choose to duplicate the file. When you choose this option, Sherlock opens the **File Properties** dialog where you can specify the name of the new file and save it to the **Project Tree**.
- Updated gRPC support to version 1.60.0.
- The Sherlock application can now process multiple board data sets when importing a .tmap file. Board names in the .tmap file no longer have to match CCA board names in the Sherlock project. If one or more board names do not match, Sherlock launches a dialog where you can assign each board data set to the correct CCA.
- When importing Ansys Mechanical .rst result files, the Sherlock application now associates materials to elements directly. You no longer have to assign elements to specific Named Selections for Sherlock to obtain the material properties.
- You can now add a temperature offset value for Icepak .tmap files.
- Added the Ball Channel Width formula to the help text in the Parts Editor.
- Updated the Parts List so it now highlights any selected rows when the Advanced Setting **Show Parts List Sources** is disabled.

3.4. Other Improvements

- Sherlock now participates in the Ansys Product Improvement Program. See [The Ansys Product Improvement Program in the Sherlock User's Guide](#).
- Added new materials: ALUMINUM NITRIDE, SILICON CARBIDE, and SILICON NITRIDE.
- Added new solders: LF35, SACQ, and 62SN36PB2AG.
- Removed the ability to export an FEA Model to Icepak (Classic), which has the .tcl extension.
- Updated the thermal conductivity of Silicon from 15.6 W/mK to 148 W/mK.

- Corrected the Generic Flex Polyimide laminate CTE_x and CTE_z values, which were reversed.

4. API Updates

Links:

[4.1. New APIs](#)

[4.2. Modified APIs](#)

4.1. New APIs

- Added **listLayers()** to list all the available layers as seen in the layer viewer for a project CCA.
- Added **exportLayerImage()** to export one or more 2D Layer Viewer images from a project CCA.
- Added **updateComponentFailureMechanismProps()** to update the Component Failure Mechanism analysis properties.
- Added **getSolderInfo** to return not only solder names but other information like type and modulus values.
- Added **updateSemiconductorWearoutProps()** API to set properties for Semiconductor Wearout analysis.
- Added **updatePTHFatigueProps()** API to set properties for Plated Through Hole analysis.
- Added **updatePadProperties()** API to set PCB pad properties for one or more parts.
- Added **getPartsListProperties()** API to get properties for parts in the Parts List.
- Added **deletePartsFromPartsList()** API to remove parts from the Parts List.
- Added **importGDSIIFile()** to import a GDSII file and any optional, associated configuration files.
- Added **addOutlineFiles** API for CSV/Excel, Gerber, and IPC-2581 outline files.

4.2. Modified APIs

- Updated **addThermalMaps()** and **updateThermalMaps()** APIs to include the optional **Add Temperature Offset** parameter.
- Modified the calculation of resin properties for stackup laminate layers so that the calculated CTE_z of the composite material has an upper value limit of 1.2 times the laminate material CTE_z.

5. Bug Fixes

- Corrected the following issue with the Layer Viewer that happened in Edit Mode: When you selected various polygons and then clicked the **Close** button, the polygons would be missing in the Layer Viewer the next time you opened it.

- When creating a CCA from a modeling region, the operation sometimes failed because the layer metadata could not be generated. This issue has been fixed.
- In some cases, when the Sherlock application created STEP file components that had a lot of inside turns, the resulting component would be inside-out if the option **Heal bodies and faces** was not selected when imported into the Ansys Discovery or Workbench applications. This bug has been corrected.
- The Layer Viewer now shows the correct component colors when displaying the strain map layer generated from a CSV file.
- When you crop image files to a modeling region, the Sherlock application no longer assumes the board is centered on the origin.
- The Exy value for the laminate Hitachi FR-4 MCL-E-700G has been corrected.
- Corrected the grammar in the tooltip for the Analysis Property **Harmonic Vibe Result Count**.
- Minimum value warnings for stub leads now appear in the console during model export operations.
- Corrected a bug in **addShockProfiles()** API. It returned an error in the response when the API called for a Phase and Event name for a non-shock Event in the project.
- Fixed an issue where 3D models with potting regions were generated with missing elements.
- When adding a .tmap file to a CCA that has data for a subassembly, after the file is created for the subassembly, the Sherlock application now sets the file type and creates thermal maps from the file.
- Corrected an errant warning in the Solder Fatigue analysis that stated the IMS model was not the expected model for the BTC package type.
- PCB surface strain results for FEA components have been updated so that the surface with the maximum strain is selected rather than the maximum stress. This prevents problems with Layered Element and Uniform Element models which, on some surfaces, might have stresses that are higher than strains.
- Corrected an issue with imported result files containing trace mapping not generated by the Sherlock application. In some cases, the trace mapped elements were not assigned the correct Named Selections which resulted in the wrong material properties being selected.
- Fixed an issue that occurred when running a Solder Fatigue Analysis using thermal maps: In some instances, for topside thruhole parts, the Sherlock application was not checking the temperature of the bottom side of the board.
- Fixed an issue in the **updateHarmonicVibeProps()** API where it was possible to enter an invalid number for the harmonicVibeCount parameter.
- The Solder Fatigue analysis model for IMS boards no longer includes thermal shock factor effects.
- Correct an issue where the selection box would sometimes not show up in the PCB Layer Viewer.
- When you save analysis results for a CCA with subassemblies, the results for each subassembly now appear in the **Project Tree** under **Saved Results**.

- Fixed not setting the part editor properties for a CC package when adding a new part to the parts list and choosing a CC package from the package chooser.
- When creating a CC package, you no longer get an "invalid units" exception if you haven't set the **Package Units** in the Parts List or Part Library.
- Package Thickness no longer goes missing when you add a new part to the Parts List after having used the Package Chooser or having used multi-part editing.
- You no longer have to restart the Sherlock application to run an FE analysis after renaming a project.
- You can now leave the Package Thickness property blank in the Parts Editor when the field is not required.
- When exporting archived results, in some cases the results files for each CCA were not being exported. This has been corrected.
- Image layers in the Layer Viewer are now displayed using the correct color as assigned in Color Settings for that layer.
- The IMS solder model has been corrected for the FR4 stackup: The substrate thickness is now pulled from the total stackup thickness minus the thickness of the two layers nearest the component. Also, the material properties for the overall stackup are used for the substrate material properties.
- For laminate layers in a stackup, the calculated CTEz value of the composite material is now set to 1.2 times the CTEz value of the laminate material.
- The Thermal Mech analysis no longer throws the error message for gullwing leads that the lead bend is not between 45 and 90 degrees when Advanced Lead Meshing is disabled.
- Corrected the issue where overlapping parts did not return the correct part temperature from CSV thermal maps.
- The Import Part Data option no longer fails for XML files exported from Sherlock where the frType parameter was equal to N/A.
- In the Package Manager, fields in the **Package** tab now display tool tips.
- Corrected the issue of project materials being loaded into the material libraries (Laminate, Material, and Solders Libraries) when starting the Sherlock application in standard mode.
- When you create a Detailed BGA Component, stackup material properties now reflect the correct values immediately.
- When editing the stackup of a Detailed BGA Component, the stackup thickness would be incorrect if the stackup unit was not set to **mil**. This has been corrected.
- When multi-editing packages in the Package Manager, updated values are now displayed immediately for all edited packages.
- LCCC packages without pad pitch defined no longer cause an error when running a Solder Fatigue analysis.
- The Semiconductor Wearout analysis now ignores parts that have a die material other than silicon.

- In some cases, when an analysis produced only errors, analysis item context menus did not enable the **View Results** menu item. This has been corrected.
- When importing CSV or Excel files which contain specifications for a polyshape, the Sherlock application now performs *case-insensitive* matches against imported values.
- Corrected the calculation of composite CTE material properties for the CGA.
- Increased the size of the **Layer** column in Stackup table to prevent the column header from being truncated.
- Fixed an issue that occurred when editing ICT Fixture spreadsheet files: X and Y values were ignored due to a case-insensitive column check, causing the lower case x and y values to be used instead. The column check is now case-sensitive.
- You can now use the Sherlock-Workbench workflow on Linux even if the application is not installed as the root user.
- In **Sherlock Settings > Data Store**, several data store file options were available that were unused by the Sherlock application—such as Package Wizard file, Failure Rate file, and Failure Class file. These options have been removed.
- Corrected a malformed error message that can be returned from the API when exporting a trace model.

6. Third Party Dependency Changes

- Updated sqlite-jdbc to version 3.48.0.0.
- Updated xercesImpl to version 2.12.2.
- Updated commons-compress to version 1.27.1.
- Updated commons-configuration2 to version 2.11.0.
- Updated commons-logging to version 1.3.4.
- Updated commons-text to version 1.13.0.
- Updated commons-io to version 2.18.0.
- Updated commons-lang3 to version 3.17.0.
- Updated Apache POI to version 5.4.0.
- Updated SparseBitSet to version 1.3.
- Updated log4j-api to version 2.24.3.
- Updated xmlbeans to version 5.3.0.
- Updated Apache PDFBox to version 2.0.33.
- Updated slf4j-api and slf4j-simple to version 2.0.16.

- Updated animal-sniffer-annotations to version 1.23.
- Updated checker-qual to version 3.33.0.
- Updated error_prone_annotations to version 2.20.0
- Updated grpc to version 1.60.0.
- Updated gson to 2.10.1.
- Updated guava to version 32.0.1-jre.
- Updated j2objc-annotations to version 2.8.
- Updated perfmark-api to version 0.26.0.
- Updated protobuf-java to version 3.25.6.
- Updated proto-google-common-protos to version 2.22.0.
- Updated image-io to version 3.12.0.

Forming

As part of the 2025 R2 Forming release, you gain access two new features, One Step and Reflection Lines, as well as enhancements to the Material Library and the Drawbeads and Mesh Check features. You can also test new beta features.

- 1. [New Features and Enhancements](#)
- 2. [Beta Features](#)

1. New Features and Enhancements

New Features

- **One Step**, a new project type, is a rapid forming simulation technique which uses the final part geometry to estimate the size of the flat blank and evaluate formability. One Step is best used in the early stages of product and process development to identify potential issues, like splits or wrinkles, before you invest in detailed tooling design.
- **Reflection Lines** (also known as zebra stripes) is a new postprocessing function in the **Analysis** tab that allows you to easily identify defects in surface quality, like wrinkles or sudden changes in curvature.

Enhancements

- **Drawbeads:**
 - Calculate the restraining and uplift forces of a 2-D drawbead using a new method for fast and accurate prediction.
 - Access three new bead profiles to support additional use cases.
 - Set bead height at zero, allowing you to exclude drawbead from your model without changing the geometry.
- **Material Library:**
 - Access material data from ThyssenKrupp and Voestalpine.

Note:

You cannot modify material data from these sources.

- Interact with an updated kinematic hardening material model in the **Material Editor**.

→ For materials using the **Barlat'89** or **Barlat'2000** model, we implemented kinematic hardening according to the Chaboche-Roussilier model. Review the [LS-DYNA Keyword Manual, Vol. II, MAT_036](#) for information about how to calculate CRCn and CRAn values.

→ We also implemented support for the **Vegter** model (**MAT_136**). To use the model, you must input Normalized Yield Stress (**FUN**), First Normalized Yield Stress for Plain Strain Test (**FPS1**), Normalized Position of Second Principal Stress Under Plain Strain (**ALPS**), and Shear Stress (**FSH**) for 0°, 45°, and 90°.

You may also define the distribution of hardening used in your curve fitting. The parameter Alpha = 0 represents pure kinematic hardening while Alpha = 1 represents pure isotropic hardening.

- When using the **Barlat'2000** model, you may now input **From Test Data** instead of alpha values.

Input the following test data and click the **calculator** icon. The application completes the curve fitting and updates the Yield Locus plot.

Required test data:

- Yield Stress (0°, 45°, and 90° directions)
- XX, YY, and XY Components of Stress on Yield Surface
- R Value(0°, 45°, and 90° directions)
- XX, YY, and XY Components to Yield Surface

- [Mesh Check](#):

- You can now choose to **Show Critical Vertices** from the Mesh Check task panel, a feature that highlights boundary vertices with exceptionally high curvature. This feature allows you to quickly identify potential issues along boundaries, refine your mesh more efficiently, and gain deeper insight into regions prone to defects like thinning, wrinkling, or cracking.
- Mesh Check now displays the **Outer Boundary** of your part. This feature helps ensure the outer contour is clean and simulation-ready, improving accuracy in tool contact, material flow, and overall mesh integrity
- You can now perform a Mesh Check on adaptive meshes.

Note:

Adaptive constraints can interfere with the automatic repair process. If automatic repair fails in areas with adaptive constraints, you may need to repair issues manually.

2. Beta Features

Beta features include Solid elements for blanks, tailor-welded blanks, non-linear friction, and a new forming material (*Mat_320) to increase overall simulation efficiency.

Note:

Beta Features Disclaimer

This is beta documentation for one or more beta software features.

- Beta features are considered unreleased and have not been fully tested nor fully validated. The results are not guaranteed by Ansys, Inc. (Ansys) to be correct. You assume the risk of using beta features.
- At its discretion, Ansys may release, change, or withdraw beta features in future revisions.
- Beta features are not subject to the Ansys Class 3 error reporting system. Ansys makes no commitment to resolve defects reported against beta features; however, your feedback will help us improve the quality of the product.
- Ansys does not guarantee that database and/or input files used with beta features will run successfully from version to version of the software, nor with the final released version of the features. You may need to modify the database and/or input files before running them on other versions.
- Documentation for beta features is called beta documentation, and it may not be written to the same standard as documentation for released features. Beta documentation may not be complete at the time of product release. At its discretion, Ansys may add, change, or delete beta documentation at any time.

Solid Elements for Blanks

You may now use solid elements instead of shell elements for your blank. Using a solid element can improve the accuracy of your simulation, especially simulations involving bending or those involving a thicker blank. The feature also allows you to better visualize through-thickness stress and strain and material flow in complex forming simulations.

1. From the **Blank** tab in the **Design-n** task panel, click **Use Solid Blank**.
2. Input the number of layers you will see in the graphics window.

Considerations for solid blanks:

- Solid blanks currently only support the Barlat'2000 yield function (**MAT_133**).
- While this feature is in development, we recommend using blanks with a geometrical tool surface offset already defined for the most consistent results.

Tailor-Welded Blanks

In some cases, manufacturers may weld together two or more metal sheets to optimize strength, weight, and material usage during forming. Ansys Forming now supports multiple weld lines, adaptive refinement along welds, adaptive meshes, and meshes with internal boundaries.

1. From the **Blank** tab in the **Design-n** task panel, click the arrow next to the **Create Blank Component** icon (plus sign) and select **Weld Blank**.
2. In the **Define by Points** task panel, click **By Line** or **By Points** depending on your preference and then click **Select Splitting Part**.
3. Select your part(s) from the graphics window or feature tree, and click **Close**.
4. Under **Pick Options**, click **By Node** or **By Point** and split your part.
5. Click **Close**. The application displays your newly created blank segments as separate entities in the **Design-n** task panel.
6. Click **No Mat.** to define the material for each segment.

Non-Linear Friction

In sheet metal forming simulation, non-linear friction refers to a friction model where the friction coefficient is not constant but varies based on local conditions. If you select, import, or define a material model with non-linear friction, you may now input pressure, velocity, and temperature-dependent friction data inside the material card.

To access this feature, from the **Material Viewer** or **Material Editor**, in the **Material Type** column, click **Other Properties** and then click the **Friction**.

*MAT_320

You can now access a new material model, ***MAT_320**. Use this model to improve the performance of plasticity models while generalizing the input.

The parameters IPHI and ITHETA, stored as spherical coordinates (ϕ, θ), define the discretization of the yield surface. IPHI determines the number of angular divisions in the ϕ direction while ITHETA determines that in the θ direction.

Currently ***MAT_320** is supported for following materials:

Table 1: Materials Supported by *MAT_320

MAT_36	*MAT_3-PARAMETER_BARLAT	Barlat'89
MAT_37	*MAT_TRANSVERSELY_ANISOTROPIC_ELASTIC_PLASTIC	Hill'48
MAT_122	*MAT_HILL_3R	Hill'48
MAT_133	*MAT_BARLAT_YLD2000	Barlat'2000
MAT_136	*MAT_VEGTER_STANDARD	Vegter Standard

Motion Release Notes

The following sections present the new features, enhancements and changes for the Ansys Motion™ standalone Solver, Preprocessor and Postprocessor for the 2025 R2 release.

For Motion in Workbench, see the [Mechanical application Release Notes](#).

This release of Motion contains all the capabilities from previous releases plus many new features and enhancements.

- 1. [Changes in Product Behavior](#)
- 2. [New Features](#)
- 3. [Functional Enhancements](#)

1. Changes in Product Behavior

Release 2025 R2 includes several new features and enhancements that result in product behaviors that differ from previous releases. These behavior changes are presented below.

- The Pacejka tire model is no longer supported.

2. New Features

The following new features are available:

- New camera controls have been added to **Animation View** in the Motion Postprocessor to allow target tracking in translational and rotational directions.
- **Numerical Data** import in the Motion Postprocessor has been enhanced to allow import of RPC-III files.
- The Motion Preprocessor now supports the option to save simulation results after co-simulation using the Functional Mock-up Interface (**FMI**).
- A **Replace FE Body** function has been incorporated into the Motion Preprocessor to allow replacement of an existing FE Body with a new one. If the new body contains an entity with a previously-existing name, the connector linked to this entity will be maintained and connected to the entity in the new body.

3. Functional Enhancements

The following functional enhancements are available:

- The 2025 R2 release features much improved compatibility of legacy models with the new Parasolid graphics kernel.
- Finite Element (FE) data import in the Motion Preprocessor now includes the ability to filter only Rigid Body Elements (RBEs) that include Static Correction Mode. This simplifies the handling of FE data by showing only the RBEs related to connectors.
- Co-simulation with Maxwell in the Motion Preprocessor now allows selection of the specific version to be used.
- A more accurate mass and moment of inertia (MOI) value is displayed in the preprocessor dialog after a body eigenvalue analysis using the Motion solver.
- Improved multithreading yields 5 - 10 % faster velocity and acceleration constraint evaluation in models where modal bodies have numerous connectors.

LS-DYNA Release Notes

This document contains release notes for most of the documentation found on the LS-DYNA product page on the Ansys Help site.

- For release notes that are specific to the version of the LS-DYNA solver included in the unified installer, see the release notes at the beginning of the Keyword Manual Volume I found [here](#).
- For release notes that are specific to changes in the LS-DYNA systems in the Mechanical Application, see the release notes [here](#).

This release contains all of the capabilities from previous releases plus many new features and enhancements. Areas where you will find changes and new capabilities include the following:

1. New Documents

1. New Documents

The following new documents have been added to the LS-DYNA product documentation set.

- **Envyo User's Guide**

Describes the use and features of Envyo, an LS-DYNA product tool, which transfers simulation data between different meshes.

- **LS-DYNA Implicit Analysis Guide**

Describes how to set up various types of implicit analyses using the LS-DYNA solver.

Ansys Fluids Products

As part of our commitment to deliver a simplified, modern license experience, Ansys is rolling out updated license increments to our customers beginning January 2020. Ansys is making these changes so the Ansys License Interconnect and License Preferences can be eliminated, greatly simplifying the customer licensing experience. The new license files are fully compatible with previous versions of Ansys back to release 17.0 when used with Ansys License Manager 2020 R1 or later.

Release notes are available for the following Ansys Fluids products:

[Fluent \(p. 109\)](#)

[CFX \(p. 127\)](#)

[CFD-Post \(p. 129\)](#)

[TurboSystem \(p. 131\)](#)

[TurboGrid \(p. 133\)](#)

[Ansys BladeModeler \(p. 135\)](#)

[Polyflow Classic \(p. 137\)](#)

[Forte \(p. 139\)](#)

[Ansys Chemkin \(p. 141\)](#)

[FENSAP-ICE \(p. 143\)](#)

[EnSight \(p. 147\)](#)

[Rocky \(p. 155\)](#)

[Freeflow \(p. 151\)](#)

Fluent Release Notes

The following sections contain release information for Ansys Fluent 2025 R2.

1. [Supported Platforms for Ansys Fluent 2025 R2](#)
2. [New Features in Ansys Fluent 2025 R2](#)
3. [Updates Affecting Ansys Fluent 2025 R2 Code Behavior](#)
4. [Features To Be Removed in a Future Release](#)

Backwards Compatibility: Ansys Fluent 2025 R2 can generally read case files and data files from all past Fluent releases. Solver and model settings from previous case files are typically respected. However, in some cases due to defect fixes and core improvements to improve robustness and/or performance, convergence behavior and/or results obtained may be different. Such release-to-release changes are documented in the Fluent Migration Manual, along with instructions to recover the previous behavior when possible.

1. Supported Platforms for Ansys Fluent 2025 R2

Information about past, present, and future operating system and platform support is viewable via the [Ansys website](#).

2. New Features in Ansys Fluent 2025 R2

The following sections list the new features available in Ansys Fluent:

- 2.1. [Meshing Mode](#)
- 2.2. [Solution Mode](#)
- 2.3. [Fluent Workspace Applications](#)

2.1. Meshing Mode

New features available in the meshing mode of Ansys Fluent 2025 R2 are listed below.

Fluent Guided Meshing Workflows

Mesh Generation

- The Rapid Octree mesher is now improved in the following ways:
 - When creating a volumetric refinement region for the Rapid Octree mesher, the ability to define its extents and orientation using a body of influence (BOI) is now supported as a full feature. A BOI is an arbitrary closed surface that you have previously imported or created, and may represent your intended refinement region better than the other options (that is, a box or frustum). ([Refinement Regions](#))

- When using the boundary projection treatment method with the Rapid Octree mesher, the ability to select a discrete mesh optimization (DMO) scheme for boundary mesh optimization is now supported as a full feature. The DMO scheme can result in significantly less defeaturing compared to the scheme available in previous releases (which is still available, and is now referred to as the conjugate gradient scheme). ([Boundary Mesh Optimization](#))
- When using the boundary projection treatment method with the Rapid Octree mesher, the ability to specify surface sizing definitions that have proximity refinement settings is now supported as a full feature. Such proximity refinement settings may be useful when you have gaps and smaller openings, as it can avoid the unwanted closing of flow paths due to defeaturing. ([Custom Boundary Sizes](#))
- When specifying the volume to be meshed by the Rapid Octree mesher, you can now use generic region definitions, each of which allow you to specify a material point and the associated face zones that are contained in or bound the volume. Such definitions can be helpful when you have a geometry where face zones overlap or should be discarded (which can introduce ambiguity in the boundary face zone assignment) or for a case where only a smaller sub-volume of a complex model is to be meshed. ([Specifying the Volume](#))
- When defining the geometry for the Rapid Octree mesher, you can now specify a reference size, which is used to expand the currently defined bounding box and thereby adjusts what cell sizes are possible in the mesh. This can be a more convenient way to precisely adjust the cell size (compared to manually changing the bounding box extents), and is especially helpful when the geometry that you input has exterior boundaries (for an interior flow problem, for example). ([Defining the Reference Size](#))

2.2. Solution Mode

New features available in the solution mode of Ansys Fluent 2025 R2 are listed below. Where appropriate, references to the relevant section in the User's Guide are provided.

Licenses

- A new capability level named CFD HPC Ultimate is available for the Fluent CPU or GPU solver. This license includes the capabilities of the CFD Enterprise license (as described in [Program Capabilities](#)) and also allows the solvers to run on any number of CPU cores or GPUs without using additional HPC licenses. It can be selected in Fluent Launcher (as described in [Selecting the Licensing Level](#)) or with the command line option `-license=hpc_ultimate`.

The CFD HPC Ultimate license was first introduced in service pack 2025 R1.01, and is now improved in the following ways:

- Aero, Icing, and Polyflow workspaces are now supported with the CFD HPC Ultimate license.
- Concurrent parametric workflows in Fluent are now supported with the CFD HPC Ultimate license.

User Experience

- Access to Ansys Engineering Copilot ([Ansys Engineering Copilot User's Guide](#)) is added to the **Learning and Support** ribbon tab.

- You can now submit batch jobs using the Fluent Launcher. Refer to [Setting General Options in Fluent Launcher in the *Fluent User's Guide*](#) for additional information.

Files

- You can save 3D data (pictures) in USD format for use in NVIDIA Omniverse. Refer to [Using the Save Picture Dialog Box in the *Fluent User's Guide*](#) for additional information.
- You can write case settings corresponding to the **Results** node in the Outline View tree to .txt and HTML format. Refer to [Exporting, Copying, and Importing Case Settings in TSV, Text, HTML Format in the *Fluent User's Guide*](#) for additional information.
- The third-party software VKI is upgraded to Ceetron-SAM 2.1.0 to address security concerns for file import/export.
- CGNS is updated to version 4.5.
- Transient data export to EnSight DVS format now automatically includes all defined report definitions. Residuals data is also available.
- The user interface is now improved for reading the lightweight data of a case or mesh file, that is, reading only the case settings of the file without any of the data associated with the mesh. When setting up a case that has a large cell count, this allows you to review and/or modify the settings without having to wait for the reading of the mesh, on a machine that does not have the memory needed for the mesh (such as a personal computer or laptop).

You can also now display surface meshes in the graphics window when you have read lightweight data. This can be helpful when setting up boundary conditions, monitors, animations, and so on.

For further details on reading lightweight data, see [Reading the Lightweight Data of Mesh or Case Files](#).

Solver-Meshing

- You can now easily disconnect one or more cell zones from all adjacent cell zones to which they are connected through shared faces and/or nodes, so that the cell zones can move relative to each other (for example, as part of a sliding mesh or dynamic mesh simulation). ([Disconnecting Cell Zones](#))
- When using a dynamic mesh zone in a simulation that models ablation or erosion / accretion, the handling of sharp corners in the mesh is now improved, which may improve performance, accuracy and robustness. For details about ablation or erosion / accretion, see [The Ablation Condition at Wall Boundaries](#) and [Particle Erosion Coupled with Dynamic Meshes](#), respectively.
- For overset meshes, you can now manually identify component meshes as collar meshes, so that they are sure to be protected from being cut by any other boundary during the hole cutting step. This can be useful in instances when the automatic treatment of overlaps is insufficient and when other hole cutting control methods are not as convenient to set up. ([Defining Collar Meshes](#))
- When creating non-conformal mesh interfaces, the text command for enabling adjustable tolerances (define/mesh-interfaces/auto-options/set-one-to-one-pairing-tolerance) is improved. You now have the option of specifying an absolute tolerance, so

that you can directly define the distance that is used in the pairing of boundary zones for mesh interfaces (rather than having to specify a relative length factor that is then multiplied by the largest mesh length scale of the interface boundaries). This provides you with greater control, and may allow you to eliminate unnecessary mesh interfaces or produce missing mesh interfaces. ([Using a Non-Conformal Mesh in Ansys Fluent](#))

Cell Zones and Boundary Conditions

- The perforated wall model is now supported with the density-based solver.
- The following improvements are now available to assist with the proper setup of adjacent cell zones with different materials:
 - When changing the material of a cell zone, a warning is now printed in the console if an interior boundary zone is detected between that cell zone and an adjacent cell zone with a different material but the same type (fluid or solid).
 - During a mesh check, the check now fails if an interior boundary zone is detected between cell zones of different materials, and (if the `mesh/check-verbosity` text command is set to 3) a warning is now printed in the console. Note that the check will not fail if the case uses the battery model, heat exchanger model, or the solid oxide fuel cells (SOFC) model, as these models allow interior zones between different materials.
 - It is now easy to slit all interior boundary zones that are between not only solid but also fluid cell zones of different materials to create coupled wall / wall-shadow pairs, through the `mesh/modify-zones/slit-interior-between-diff-materials` text command.
 - In previous releases, when you fuse two uncoupled face zones (for example, an uncoupled wall / wall-shadow pair), this always produces a single face zone of type interior. Now the Fluent software checks during the fusing operation if the adjacent cell zones are of different materials or material types, and if they are, automatically slits the resulting interior zone into a coupled wall / wall-shadow pair, so that it is no longer necessary to slit it manually.

Materials

- The non-Newtonian Carreau viscosity model now uses a more general formulation of the Yasuda-Carreau model, which includes a shape parameter that controls the transition of the viscosity profile between zero-shear and infinite-shear viscosities. ([Power Law for Non-Newtonian Viscosity](#))

Moving Reference Frames

- For cases that have one or more moving reference frame zones, the ability to use a time-step-size-independent continuity discretization is now fully supported. Such discretization reduces the degree to which the results are dependent on the time step size for transient simulations that use frame motion zones with the pressure-based solver. ([Solution Strategies for a Single Moving Reference Frame](#))

Turbulence

- Optimized LES Numerics are now available for the GPU solver, which typically allows for a reduction in the number of iterations used per time step. ([Optimized LES Numerics](#))

- For the **GEKO** model with the GPU solver, the **Wall Distance Free** formulation is now supported. For details, see [Generalized \$k-\omega\$ \(GEKO\) Model](#).
- The **Viscous Heating** option label in the **Viscous Model** dialog box is now changed to **Viscous Work**, in order to better reflect the effect of this option.

Turbo Setup Workflow

- Fluent's Turbo Workflow has been renamed to the Turbo Setup Workflow. For more information, see [Using the Turbomachinery Setup Guided Workflow](#).
- Fluent's Turbo Workflow has been enhanced such that, if a completed setup is subsequently reopened, you can choose to hide or disable the Workflow tab in the user interface. Additionally, if there are no associated workflow files available, then the Workflow tab is now hidden by default. For more information, see [Using the Turbomachinery Setup Guided Workflow](#).

Turbomachinery

- When Turbo Models is enabled, the Dynamic Mesh Zones dialog box now allows you to select one or more cell zones to create multiple dynamic mesh zones of the same type (for example, blades) simultaneously. ([Creating and Applying Dynamic Mesh Zones](#))

Reacting Flows

- The dual-potential cell zone condition is now available for modeling electrochemical reaction in a porous medium. ([Electrochemical Reaction in Porous Medium](#))

Discrete Phase Model

- For contour plots of DPM particle sampling on a planar surface, the `<surfname>_signif_conc_estimate` and `<surfname>_signif_conc_time_integral` expressions have been added to evaluate the local discrete phase concentration. ([Contour Plots of DPM Particle Sampling Results on a Planar Surface](#))

Multiphase Models

- An enhancement to the source term linearization for the Wet Steam model is available to improve robustness and convergence speed in some cases. ([Setting Up the Wet Steam Model](#))
- The quadrature-based moment method (QBMM) has been released as a full feature. The method can be helpful for cases where a higher order of accuracy for solving population balance equations is preferred. This was a beta feature in previous release. ([The Quadrature-Based Moments Method \(QBMM\)](#))
- The Lee evaporation/condensation model now supports a **user-defined** method for specifying the **From Phase Frequency** and **To Phase Frequency** model parameters using a `DEFINE_MASS_TR_PROPERTY` user-defined function (UDF).
- For the mixture multiphase simulations that involve the semi-mechanistic boiling model, the following enhancements have been implemented ([Using the Semi-Mechanistic Boiling Model](#)):
 - You can now specify **Vapor Heat Transfer Coefficient** for the single phase heat flux. For clarity, the previous **Heat Transfer Coefficient** parameter has been renamed to **Liquid Heat Transfer Coefficient**.

- When the semi-mechanistic boiling model is used, the **Boiling Model Expert Options** are now always visible in the **Evaporation-Condensation Model** dialog box. In previous releases, they were displayed by using the text user interface (TUI).
- The **Critical Liquid Volume Fraction** is now limited by a maximum value of 0.65.

Eulerian Wall Film Model

- Film mass and heat fluxes through source terms are now include in the evaluation of the film mass flow rate and film heat transfer rate in the film flux reports. Film heat source term input now must be in the form of $mC_p(T - T_{ref})$ (here, m is the film mass, C_p is the film liquid specific heat, T is the film temperature, and T_{ref} is the reference temperature of 298.15K).

Electric Potential Field and Electrochemistry Model

- The swelling model is now available for the resolved lithium-ion battery model. ([Lithium-ion Battery Swelling Model in the Fluent Theory Guide](#))
- For the electrolysis and H2 pump model, the following features have been added:
 - A new device, PEM fuel cell, is now available. This device can be used for modeling PEM fuel cell involving multiphase flows. ([Electrolysis and H2 Pump Model](#))
 - A new device, high temperature PEM electrolysis, is now available. This device is suitable for modeling single-phase electrolyzers that operate at high temperatures. ([Electrolysis and H2 Pump Model](#))
 - The unresolved 1D modeling approach has been added. This approach is an extension of the existing unresolved 0D approach and accounts for water transport across the unresolved membrane electrode assemblies (MEA) zones. ([Unresolved 1D Modeling Approach](#))

The swelling model is now available for the resolved lithium-ion battery model. ([Lithium-ion Battery Swelling Model in the Fluent Theory Guide](#))

Battery Model

- A new, more general thermal abuse model, called N-equation kinetics model, has been added. This model incorporates a user-specified number of user-defined thermal abuse reactions. ([General N-Equation Model](#))

Solver Initialization

- A text command is now available to flush the profile memory for a boundary. In some cases with solution-dependent profiles on boundary zones, re-initialization may not remove all data from the boundary and a complete flush may be needed. ([Flushing Profile Memory on Boundary Zones](#))
- When using the pressure-based solver with a turbulence model, you can now enable an alternative treatment option, such that the effects of turbulence wall functions are removed at walls where you have specified zero shear. This treatment may enhance convergence and solution behavior. ([Specified Shear](#))

Fluent Native GPU Solver

- It is now possible to enable many-to-many CPU/GPU remapping for the native GPU solver using Fluent Launcher. ([CPU/GPU Remapping](#))
- The exporting of solution data calculated by the native GPU solver as EnSight DVS files for postprocessing in EnSight is now supported as a full feature. As part of the export, you can specify the file name, cell zones and/or surfaces, quantities, and so on, similar to when exporting from the CPU solver. Exporting after a calculation (either steady or transient) or during a transient calculation is supported. ([Exporting Solution Data as EnSight DVS Files](#))
- Cylindrical profiles are now supported for cell zones and boundary conditions.
- The 3D fan cell zone condition is now supported.
- The following boundary conditions are now supported:
 - Inlet Vent
 - Exhaust Fan
- GPU Solver performance improvements have been made for the following models:
 - S2S model with non-changing geometries/boundary conditions
 - Sliding mesh
 - Flamelet Generated Manifold (FGM)
- Modeling electric potential is now supported.
- You can now track the current and peak memory of GPUs.
- Viscous work is now supported for applicable turbulence models.
- The ability to enable asynchronous outputting for the native GPU solver is now supported as a full feature. This can improve the solver performance during the calculation (with the speed increasing by up to a factor of 2) when postprocessing monitors, rendering animations, auto-saving case / data files, and/or automatically exporting data. ([Asynchronous Outputting](#)).
- The use of expressions for report definitions and unsteady statistics with the GPU solver is now supported as a full feature for fully supported models. For a list of supported fields and other details, see [Using Expressions with Report Definitions and Unsteady Statistics](#).
- The Discrete Phase Model (DPM) features supported with the GPU solver have been upgraded from beta to release. For details, refer to the following sections:
 - [Discrete Phase Model](#)
 - [Boundary Conditions for the DPM](#)
 - [Limitations for the DPM](#)
 - [Solver Settings for the DPM](#)

- [Postprocessing for the DPM](#)

Density-based Solver

- The compatibility between the two-temperature model and CHEMKIN mechanism import has been enhanced. This enhancement allows you to import CHEMKIN format kinetic mechanism, thermodynamic database, and transport property database into the two-temperature model. Additionally, the issue of Blottner curve fit coefficients being incorrectly set to zero during the CHEMKIN import has been addressed.

Graphics, Reporting, and Postprocessing

- Annotations are now associated with graphics objects (mesh, contours, vectors, pathlines, particle tracks, LIC plots) once they are added to the display. This means on re-display, any associated annotations are displayed along with the graphics object. Refer to [Adding Text to the Graphics Window in the Fluent User's Guide](#) for additional information.
- The ranges for X and Y axes on 2D plots, such as XY Plots and Histograms, can now be controlled individually, allowing you to manually limit one range while having the other computed automatically.
- You now have the option to manually specify the increments of major and minor gridlines in 2D plots. Refer to [Controlling the Major and Minor Gridlines in the Fluent User's Guide](#) for additional information.
- The ranges for X and Y axes on 2D plots, such as XY Plots and Histograms, can now be controlled individually, allowing you to manually limit one range while having the other computed automatically.
- When printing the memory usage to the console in Windows (as described in [Memory Information](#)), the resident memory will now be printed along with the virtual memory by node and by host. Note that for Windows, the virtual memory corresponds to private bytes and the resident memory corresponds to the working set; for further details, see the Microsoft online documentation.
- A preference for surface caching is added to improve performance of mesh redisplay and incremental display, even across mesh objects. This is beneficial for cases of all sizes. Refer to [Graphics Performance in the Fluent User's Guide](#) for additional information.
- The preference for improving the display and interactive performance of mesh surfaces is extended to the global mesh display. The preference is **Optimize Input Data** in the **Graphics** branch of Preferences (**File > Preferences...**).
- A new **Viscous Dissipation** field variable (in the **Temperature...** category) is now available when the energy equation is enabled and the **Viscous Work** option is enabled in the **Viscous Model** dialog box.

Parallel Processing

- The `-platform=<x>` command line option (which is available only on x86 Linux, and can enhance performance when running on processors that support the AVX2 instruction set) is improved:

- The `<x>=intel` setting is now more reliable. With this setting, AVX2 binaries are used for the host and node processes, along with the Intel MKL sparse LDU smoother.
- The `<x>=amd` setting is now fully supported. With this setting, AVX2 binaries are used for the host and node processes.
- When partitioning the mesh for a simulation with multiple processes or GPUs and the Laminar, EDC, or PDF transport model, you can now enable a Stiff Chemistry weighting to more evenly distribute the computational load corresponding to the reaction integration. ([Partitioning](#))

Adjoint Solver Module

- The changes listed below have been made to the design tool settings.

In the Design Change tab:

- Morphing method selection has been changed to a drop-down list.
- The constraint method is now specified by either enabling or disabling the Enhanced Constraint Satisfaction option, for the enhanced constraint method and standard constraint method, respectively.
- The default morphing method has been changed from Polynomials with the standard constraint method to Polynomials with the enhanced constraint method.
- Freeform Scaling Scheme settings are now hidden unless the Polynomials morphing method with the standard constraint method is specified.
- The Applied Conditions list has been renamed to Applied Constraints.

In the Region Conditions tab:

- The Invariant option has been renamed to Shape Preserving.
- The design tool settings visibility option has been added to the design tool which allows for either basic settings or advanced settings to be exposed.

When basic is selected for the design tool settings visibility, the settings listed below will be hidden.

Design Change settings:

- Morphing method
- Fix surfaces...

Objectives settings:

- Increase Value and Decrease Value

Region Conditions settings:

- Continuity Order and Definition options for Region Boundary Continuity

Numerics settings:

- All Numerics settings are hidden
- When Advanced is selected for design tool settings visibility, the following Numerics settings have been changed:
 - Prescribed motion settings are now hidden when using the polynomials morphing method with the standard constraint method (can still be specified using the TUI).
 - In the Freeform Motion group box, Constraint Relaxation and Parameter Relaxation settings are now hidden (can still be specified using the TUI).
 - The Tolerances settings group box has been renamed to Residual Tolerances.

Parametric Studies

- Parametric reports exported as PowerPoint slides have various enhancements, such as an embedded Excel spreadsheet representation of the parametric table, as well as the ability to select specific filtered XY plots and slider-based images and image comparisons and add them to your exported PowerPoint presentations. For more information, see [Enhancing Exported PowerPoint Parametric Reports](#).
- You are able to access visualization tools specific for your optiSlang-generated design point study. For more information, see [Visualizing Your Parametric Study Data Using optiSlang](#).
- You can specify various ways to customize the report-related settings that will be applied while design points are updated in the parametric study. See [Automatically Setting Design Point Report Options](#) for more information.

Cloud Computing

- Signing into Ansys Cloud from within Fluent now opens a separate authentication panel within Fluent itself by default. You can choose to authenticate using a separate web browser page using the appropriate preference. See [Accessing Ansys Cloud](#) for more information about Ansys Cloud within Fluent.

System Coupling

- Rigid Body FSI is now available via System Coupling and Fluent. ([Supported Capabilities and Limitations](#))

Expressions

- You can rename named expressions, including those referenced by other expressions and those in use in boundary conditions and elsewhere. The new name of a renamed expression will automatically be updated at the dependent locations, ensuring they do not become invalid. Refer to [Named Expressions in the Fluent User's Guide](#) for additional information.

Web Interface

- Various graphics-based enhancements have been introduced, such a more interactive triad in the View Arc, as well as interactive triads when working with multiple plane surfaces (see [Creating and Displaying Multiple Plane Surfaces](#) for more information).

- You can use the **Graphics** panel to adjust the colors and associations related to displaying objects in the graphics window. See [Working With Graphics Objects and Your Simulation Results](#) for more information.
- You can play animations, view individual frames in the animation, as well as pause and resume the animation at any frame. See [Performing Calculation Activities](#) for more information.
- Several improvements have been made to panels and selection lists in order to enhance usability. See [Property Panels](#) for more information.
- For cell zone and boundary conditions, within their respective context menus and/or panels, you can choose to **Display** the surface or to **Create Surface** if one does not already exist.
- When displaying the results of your simulation, you can edit various aspects of the colormap by double-clicking it in the graphics window. See [Color Maps](#) for more information.
- You can choose a specific color for any selected object(s) in the graphics window by using the **Color by > Uniform color** option from the context menu. Use the **Esc** key to release any selection(s) in the graphics window. Subsequently, you can randomize the colors of unselected objects displayed in the graphics window using **Color by > Randomize colors** option from the context menu.
- You can set the transparency of selected objects in the graphics window using the **Set transparency** option. This displays a slider control (and a numerical input control for values greater than or equal to 0 and less than or equal to 100) where you can adjust the visibility of the selected object. Use the **Reset transparency** option to disable your previous visibility/transparency settings.
- The **View options** in the View Arc now contains a toggle between the perspective projection view and the orthographic projection view (which itself reveals the ability to optionally display a ruler in the graphics window).

Beta Features

- There are also some exciting new enhancements available as beta features that you may be interested in trying out. Detailed documentation is in the *Fluent 2025 R2 Beta Features Manual*.

2.3. Fluent Workspace Applications

New features available in the client applications of Ansys Fluent 2025 R2 are listed below.

Fluent Icing

Fluent Icing allows you to easily conduct in-flight icing simulations within a dedicated Fluent Application Client environment. The following additional functionality has been added in this release.

- **Ice Displacement and Remeshing Updates**

Multishot with remeshing workflows no longer require wall boundaries to be separated along sharp edges to capture these features properly during the wrapping process. This is a major usability upgrade, allowing users to continue the icing process without modifying their original grid boundary topologies.

The remeshing script has been relocated to the `cortex/resources/templates/prjapp/flicing/` directory and is no longer sourced from the FENSAP-ICE installation path.

Ice smoothing is now enforced at wall/symmetry plane intersections in 3D remeshing simulations to improve the robustness of the surface wrapping process and to reduce the occurrence of prism and tetrahedral meshing failures in these regions.

The **STL Ice Thickness Limit** option restricts the iced STL file to iced facets only, based on a user-defined ice thickness threshold. This provides several benefits:

- STL file size is reduced, saving storage space.
- Excessive and spotty mesh refinement caused by proximity detection between the STL and the original surface is avoided, resulting in smaller iced meshes during multishot icing.
- Remeshing time is noticeably reduced by reading a smaller STL file, minimizing proximity sizing checks, and wrapping with a more optimal sizing function.
- More optimal iced grids with reduced mesh sizes allow larger problems with more detailed ice shapes to be solved using the same computing resources.
- STLs isolated to iced areas are easier to manipulate for downstream 3D printing processes, where ice shapes can be manufactured and attached to actual aircraft surfaces.

- **User Interface, User Experience Upgrades**

The default run output file prefix is now set to `out.*`, instead of using the case file name. You can still use the case file or a custom prefix by modifying the **Output Filename** option.

Numbered file outputs now use six-digit numbering instead of two-digit, improving solution loading and animation workflows, especially in ice protection CHT simulations.

- **Non-Conformal Interface Support**

One-to-one matching mesh interfaces are now supported. These interfaces must have good overlap and be manually paired using the **Matching** option in the **Mesh Interfaces** panel.

- **CFX Turbo Import (Beta)**

A new file import mode allows conversion of multi-domain CFX `.res` files into `.cas` and `.dat` files for use in engine icing simulations. Each domain is separated into individual simulations in the project panel. You can select a subset of domains for import and rearrange their order as needed.

- **Mixing Plane Radial Profiles for DPM and Vapor (Beta)**

A mixing plane model has been developed to transfer particles and vapor between turbo rows by redistributing particle sizes into new bins based on their updated sizes and radial positions. The resulting data is stored as a radial profile in a file, which can be reused across runs and grids. The model captures changes due to wall-particle interactions such as splashing (droplets), shattering (ice crystals), and centrifugal effects. This feature enables multi-row engine icing simulations with evolving particle size distributions—functionality not supported in FENSAP-ICE-TURBO.

- **Ice Shedding with Parallel Stress Analysis and Crack Propagation (Beta)**

A new multi-CPU stress solver, **Iterative – Parallel**, has been added to the rotor ice shedding model to accelerate calculations, particularly for propeller blades using unstructured grids. The previous single-CPU solver using the LU method has been renamed **Direct – Serial**.

Fluent Aero

Fluent Aero allows you to easily explore the aerodynamic performance of aircraft under a wide range of flight regimes, from subsonic to hypersonic conditions, all within a dedicated Fluent Application Client environment. A streamlined workflow guides you through the creation of a matrix of flight conditions or design points where single and multiple flight parameters, such as angle of attack, Mach number, altitude, etc., can vary. Most common models, solvers, and convergence settings of Fluent are tuned using the latest best practices for external aerodynamic problems and are available in Fluent Aero's user interface. In this manner, simulations can be conducted in a quick and user-friendly environment. The full capabilities of the Fluent Solution workspace remain accessible when its session is displayed through **Workspaces** → **Solution** within the Fluent Aero ribbon. Tutorials are available to provide examples on how to conduct exploratory simulations using single and multiple flight parameters.

In this release, a broad range of features have been added and are described below.

- **AET: Aerodynamic Extraction Tool**

The Aerodynamic Extraction Tool (AET) has been updated to automatically extract blade sections from a 3D blade using Ansys Discovery™ instead of Ansys SpaceClaim®. Since Discovery and SpaceClaim are used as background processes of the AET, the user experience and overall workflow remains unchanged.

- **Ionized data for HFSS**

Fluent Aero now contains an ionized data converter that allows the export of point-cloud data containing permittivity and conductivity plasma properties. This converter can only be used with Fluent Aero species solutions that were obtained at high-speed flow conditions.

This data can be read by HFSS to analyze how radio frequencies are affected. It replaces an existing Python script, offering enhanced functionalities such as automatic plasma property creation for multiple solutions, optimal cloud point generation to reduce computational costs, and an interactive user interface for visualizing the plasma sheath and volume.

- **Mesh Adaption**

The **Mesh Adaption** feature using *PUMA* with the *Combined Hessian Indicator* has been moved to full release to enhance the precision and accuracy of all conditions defined inside Fluent Aero's design point table while automating the post-processing of all mesh adaption final and cycle solutions.

Management of cycle solutions generated by mesh adaption simulations have been significantly improved from its previous beta version. For instance:

- It is now possible to use the **Interrupt** and **Resume** commands to pause a cyclical solution to visualize its solution and make appropriate changes to its convergence settings.

- After completion of a simulation, it is possible to use the **Restart** and **Continue to Update** commands in a mesh adaption simulation to either improve the convergence of a given mesh adaption cycle, to increase the number of mesh adaption cycles to conduct, or to carry out more iterations to further converge a final adapted solution.

Furthermore, you can also now access the **Save Results** command to store the results of a particular cycle. You may then choose to **Run Next DP** or **Run Next Cycle (Beta)** to proceed directly to the next design point or adaptation cycle.

Post-processing of cycle solutions has been enhanced. Graphics and Scene objects can now be independently selected for image creation and saving from their respective Properties panels. A separate Results object has been introduced to control the image type and output settings. This enables saving images for the current solution (**Current Solution**), all design points (**All Design Points**), all cycles of the current design point (**Current DP - All Cycles**), or all cycles of all design points (**All Design Points - All Cycles**).

Detailed documentation on the various ways to interact with a cycle solution is available in the User Guide.

(Beta) When beta features are enabled, it is now possible to select an intermediate cycle to define as the final cycle of a particular design point calculation.

- **Aerodynamic Coefficients: Parameter Search (Beta)**

Similar to **Mesh Adaption**, improvements were made to the **Parameter Search** feature to facilitate interacting with design point cycle solutions with **Parameter Search** enabled. This includes improvements to the **Interrupt**, **Resume**, **Restart**, **Continue to Update**, **Save Results** and **Run Next DP** features.

Polyflow

The Polyflow workspace is a means to explore manufacturing applications of highly viscous materials such as polymer extrusion, blowmolding, fiber spinning, etc. within the Fluent environment.

- Support of CFD HPC Ultimate license has been added.
- Disagglomeration model allows to study the evolution of the size of solid particles in mixing flows.
- Adaptive meshing for large variation of field (beta feature) allows to refine the mesh according to the variation of a given field in finite elements.
- Surface tension for 2D and 3D free surfaces (beta feature) allows to take surface tension into account in free surfaces.
- Calibrator model (beta feature) allows to study the performance of extrusion calibrators for kinematic and thermal aspects.
- Improvement of the Polyflow Workbench Add-in (beta feature).

3. Updates Affecting Ansys Fluent 2025 R2 Code Behavior

The following sections list the code changes in Ansys Fluent 2025 R2.

3.1. Meshing Mode

3.2. Solution Mode

3.3. Fluent Workspace Applications

General

3.1. Meshing Mode

This section contains a list of code changes implemented in the meshing mode of Ansys Fluent 2025 R2 that may cause behavior and/or output that is different from the previous release.

Meshing Workflows

- Cutcell meshing, including prism generation for cutcell meshes is no longer supported.
- In previous releases, Spaceclaim (SCDM) was used as the default import route for importing .scdoc and .scdocx files. In this release, .scdoc and .scdocx files are now imported using the Discovery (DSCO) import route.
- A dynamic min area setting has been added based on the mesh scale to minimize issues if the mesh is scaled when going to the solver.

Mesh Generation

- When using the Rapid Octree mesher, the ability to set a global feature angle refinement level is removed, in order to promote clarity during setup. The ability to define such mesh refinement is now only available by creating size functions with defined angular refinement settings for one or more face zones. ([Custom Boundary Sizes](#))

3.2. Solution Mode

This section contains a list of code changes implemented in the solution mode of Ansys Fluent 2025 R2 that may cause behavior and/or results that are different from the previous release.

Files

- The requisite library for importing Abaqus ODB files is removed for Ansys Release 2025 R2 due to security concerns. Contact Ansys support if your workflow requires importing Abaqus ODB files.

Solver-Meshing

- The merging of zones is now performed faster. For further details on zone merging, see [Merging Zones](#).

Cell Zones and Boundary Conditions

- When modeling porous media using the physical velocity porous formulation, the inputs for the velocity values in the velocity inlet boundary conditions are now no longer erroneously divided by the porosity. ([Porous Media Conditions](#))

Turbulence

- The formulation of the SBES shielding function in the GPU solver has been consolidated with the Fluent-CPU solver. A term used in the original SBES shielding function to detect regions away from walls, which are not sufficiently refined for LES resolution, has been deactivated to match the Fluent-CPU behavior.

The following command can be used to reactivate the term and recover the Fluent-GPU behavior prior to the 2025 R2 release:

```
(rpsetvar 'sbes-bf-include-sub-function1? #t)
```

Reacting Flows

- For non-premixed and partially premixed combustion cases, a more accurate method has been implemented that calculates the heat release rate from the PDF table. In previous releases, the heat release rate was calculated using the cell enthalpy.

Discrete Phase Model

- When generating contour plots of DPM particle sampling, the accuracy of the calculation of the discrete phase flux through the sampling plane has been improved. If particles cross the plane in both directions, only the net flux through the individual square element of the plane is calculated. Note the following:
 - Result values may now be negative where they were positive before.
 - If only negative values are present, you can ignore the sign and use the absolute value of each data point.
 - If both positive and negative values are present, the net mass flux has different directions at different locations.
- The erosion/accretion model has been changed for cases with the **Accumulate Erosion/Accretion Rates** option is enabled in the **Sample Trajectories / Compute Erosion** dialog box. When reading a data file generated prior to 25R2 into your Fluent session, all accumulated erosion/accretion rates are discarded. After recomputing the erosion/accretion rates and saving the data file, the rates will be stored in the file and can be loaded into future sessions.

Multiphase Models

- In previous versions of Ansys Fluent, the default value of the coefficient C_{ke} in the Troshko-Hassan model formulation was 0.75, which was divided by 0.75 inside the code. The default value of C_{ke} has changed to 1, and the division has been removed. The solution results should not change compared to previous versions.
- For the VOF and mixture multiphase models, the alternative energy treatment for mass transfer has been improved resulting in more accurate solution.
- The depth function used in the TMA spectrum for random waves has been corrected to be consistent with *DNV Recommended Practice - DNV-RP-C205 - Environmental Conditions and Environmental Loads, 2019*. In previous releases of Ansys Fluent, the depth function was based on the 2010 edition of the same publication. ([TMA Spectrum](#))

Density-Based Solver

- For High Speed Numerics with the density-based solver, there are now improved relaxation and iterative parameters for transient calculations employing the implicit CFL-based solution method. These modifications increase the robustness of the inner-iterations for high-speed flows and allow improved convergence when using large implicit time-steps.

Fluent Native GPU Solver

- To ensure that you have access to the full functionality of the GPU solver on one or multiple NVIDIA GPUs, you must now install CUDA version 12.8. The supported driver version is 570 or newer, and the minimum supported hardware is the Pascal generation. ([Supported GPUs and Drivers](#))
- For transient cases with CHT solids and the solid timestepping option enabled, the treatment for solid timestepping for the CPU-based Fluent solver has been changed in this release. However, the GPU solver still uses the previous treatment.

Parallel Processing

- The resident memory required for some cases may increase approximately 25 MB per process due to third-party updates.
- The InfiniBand interconnect is now selected by default and when `auto-select` is used with Open MPI for both x86 and Arm architectures. ([Starting Parallel Ansys Fluent on a Linux System Using Command Line Options](#))

Adjoint Solver Module

- Changes to the design tool may cause journals that were created in a previous release to fail. Failure to read journals may be caused by the design tool changes listed below.
 - Journal attempting to load a case from mesh.
 - Journal attempting to load a case that was created prior to Release 23R1, which does not have the adjoint module enabled.
 - Journals may fail due to the following changes to the default morphing method, from Polynomials with the standard constraint method to Polynomials with the enhanced constraint method:
 - Journal attempts to specify polynomials-standard numerics settings.
 - Journal attempts to set freeform scaling method and factor settings.
 - Journal attempts to define increase/decrease objective type while using the polynomials-standard morphing method, such that the target change is inappropriate unless it is scaled back significantly. The polynomials-enhanced method will simply achieve the target change without scaling it back, resulting in bad deformations.
 - Journal attempting to load a case from a mesh file. The resulting deformations will not be the same between standard and enhanced constraint methods.

If a journal read failure was caused by the change to the default morphing and constraint method, adding the following line to the journal file may resolve the issue:

```
adjoint design-tool design-change select-constraint-method  
standard
```

3.3. Fluent Workspace Applications

This section contains a list of code changes implemented in the client applications of Ansys Fluent 2025 R2 that may cause behavior and/or output that is different from the previous release.

Polyflow

- – The number of processors used by the workspace solver has been increased from 2 to 4 processors.
- Mesh objects created by the workspace no longer use generic naming conventions (**mesh-1**, **mesh-2**, etc.) by default. The new default is to use the topological names found in the mesh file. The behavior can be changed using **Preferences**. See [Setting Preferences for Polyflow in the *Fluent Workspaces User's Guide*](#) for details.

4. Features To Be Removed in a Future Release

This section contains a list of features that are supported in Release 2025 R2, but will be removed in a future release of Ansys Fluent. It is recommended that you begin to migrate any cases that use these features at your earliest convenience.

Solution Mode

- **Discrete Phase Model**
 - Shared-memory parallel DPM tracking is deprecated and will be removed in a future release.

CFX Release Notes

The following sections contain release information for Release 2025 R2 of Ansys CFX.

- 1. [Supported Platforms](#)
- 2. [New Features and Enhancements](#)
- 3. [Updates Affecting Code Behavior](#)

1. Supported Platforms

Platform/OS levels that are supported in the current release are posted on the [Ansys website](#).

Note:

- If you are using CFX-Pre, CFD-Post, or TurboGrid on a remote machine and the geometry is not appearing in the viewer, try turning off the following user preference: **Common > Viewer Setup > Double Buffering**.
-

2. New Features and Enhancements

This section lists features and enhancements that are new in Release 2025 R2 of Ansys CFX.

- For multiphase flow modeling, there are two additional wall lubrication force modeling options:
 - The Hosokawa Model
For details, see [The Hosokawa Model in the CFX-Solver Theory Guide](#).
 - The Coefficient and Direction Model
For details, see [The Coefficient and Direction Model in the CFX-Solver Theory Guide](#).
- A single domain interface can now have multiple domain overlap types such as Fluid Fluid and Fluid Solid. For details, see [Heterogeneous Side Interfaces in the CFX-Solver Modeling Guide](#).
- There is a new monitor point option, Generalized Force, for cases involving time-periodic loads on walls. For turbomachinery cases, the monitoring of Generalized Force can help indicate the relative excitation of various blade deformation modes. For details on the Generalized Force option, see [\[Monitor Name\]: Option in the CFX-Pre User's Guide](#). For details on the Generalized Force calculation, see [Generalized Force Calculation in the CFX-Solver Modeling Guide](#).
- For Transient Blade Row simulations involving the Time Transformation method or Fourier Transformation method, a new expert parameter, monitor phase shift option, controls how input

data is derived for a stationary monitor point. For details, see [Transient Blade Row Model Parameters in the CFX-Solver Modeling Guide](#).

- License control options may be set using the CFX-Solver Manager and the `cfx5solve` command line utility, allowing the license capability level and HPC license to be specified on a 'per run' basis. For details, see [License Control in the CFX-Solver Manager User's Guide](#) and [Command-Line Options and Keywords for cfx5solve in the CFX-Solver Manager User's Guide](#).
- Several indirect start methods are available to support solver runs on clusters. These methods are listed here, with bold text marking the new methods:

Submit to Slurm

Submit to Slurm Consistent Reduction

Submit to Slurm Open MPI

Submit to UGE

All of these methods generate a submit script on-the-fly and, by default, wait for job completion. 'Consistent Reduction' start methods do not appear in the CFX-Solver Manager, hence are only available for command line use.

For writing your own submission script, the following direct start methods are available:

- Intel MPI Scheduler
- **Intel MPI Scheduler Consistent Reduction**
- **Open MPI Scheduler**

These methods are intended to be used without having to provide a host list; with these methods, MPI obtains a host list from the scheduler environment.

3. Updates Affecting Code Behavior

This section contains a list of changes that may cause the solution results from Ansys CFX to differ between Release 2025 R2 and Release 2025 R1.

- For injection regions with compressible flow, the **Sonic Limiter** option `Local Pressure` now includes an active forcing of the reference Mach number at each hole/slot to not exceed the target value of 1.0. Compared to the previous release, this is likely to produce more accurate results at each hole/slot, reducing any supersonic overshoot. For details, see [Flow Regime in the CFX-Pre User's Guide](#).

Also for **Sonic Limiter** cases, including cases with low mass flow such that the flow through holes/slots does not reach the target Mach number, solutions are likely to be more stable because velocities are more constrained around the edges of each hole/slot.

CFD-Post Release Notes

The following sections contain release information for Release 2025 R2 of Ansys CFD-Post.

- 1. [Supported Platforms](#)
- 2. [New Features and Enhancements](#)
- 3. [Incompatibilities](#)

1. Supported Platforms

Platform/OS levels that are supported in the current release are posted on the [Ansys website](#).

Note:

- The CFD-Post graphical user interface might not appear correctly when launching CFD-Post via VirtualGL and VNC Connect/Turbo VNC on a remote Linux platform.
 - When running CFD-Post on Ubuntu 22, if the **Report Viewer** tab is blank, try setting environment variable "QTWEBENGINE_CHROMIUM_FLAGS=-no-sandbox".
 - If you are using CFX-Pre, CFD-Post, or TurboGrid on a remote machine and the geometry is not appearing in the viewer, try turning off the following user preference: **Common > Viewer Setup > Double Buffering**.
-

2. New Features and Enhancements

This section lists features and enhancements that are new in Release 2025 R2 of Ansys CFD-Post.

Release 2025 R2 of CFD-Post has no new features or enhancements.

3. Incompatibilities

This section describes the operational changes, the procedural changes (actions that have to be done differently in this release to get an outcome available in Release 2025 R1), and the support changes (functionality that is no longer supported) in Release 2025 R2 of Ansys CFD-Post.

- Movie files of type MPEG1 can no longer be written by CFD-Post. Alternative formats, including MPEG4, can be written instead.

TurboSystem Release Notes

TurboSystem is a set of software applications and software features that help you to perform turbomachinery analyses in Ansys Workbench. For details, see [Introduction in the TurboSystem User's Guide](#).

These release notes cover:

- Turbo Setup system
- Performance Map system
- Vista AFD, Vista CCD, Vista CPD and Vista RTD
- Turbo Mesh cell
- Vista TF
- The Turbomachine Neutral Data Format

These release notes do not cover:

- Ansys BladeModeler (see [Ansys BladeModeler Release Notes](#))
- TurboGrid (see [TurboGrid Release Notes](#))
- CFX-Pre (see [CFX Release Notes](#))
- CFD-Post (see [CFD-Post Release Notes](#))

Note:

After reviewing the TurboSystem release notes, you are encouraged to see [Usage Notes](#), which describes some known TurboSystem workflow issues and recommended practices for overcoming these issues.

1. Supported Platforms

Platform/OS levels that are supported in the current release are posted on the [Ansys website](#).

2. New Features and Enhancements

This section lists features and enhancements that are new in Release 2025 R2 of TurboSystem.

- Vista TF supports roaming with a Shared Web product license.

TurboGrid Release Notes

The following sections contain release information for Release 2025 R2 of Ansys TurboGrid.

1. [Supported Platforms](#)

2. [New Features and Enhancements](#)

1. Supported Platforms

Platform/OS levels that are supported in the current release are posted on the [Ansys website](#).

Note:

- If you are using CFX-Pre, CFD-Post, or TurboGrid on a remote machine and the geometry is not appearing in the viewer, try turning off the following user preference: **Common > Viewer Setup > Double Buffering**.
-

2. New Features and Enhancements

This section lists features and enhancements that are new in Release 2025 R2 of Ansys TurboGrid.

- As part of the existing meshing process, the flow path is parameterized to obtain a spatial coordinate system that is suitable for meshing algorithms. In the previous release, the parameterization process uses a Block-structured method. In the current release, the parameterization process has a new available method: Locally-adapted.

A new user preference, Use Locally-Adapted Flowpath Parameterization By Default, controls which parameterization method (Block-structured or Locally-adapted) applies by default for new cases. For details, see [TurboGrid Options in the TurboGrid User's Guide](#).

Note that the Machine Data object's **Flowpath Parameterization Type** setting controls which parameterization method applies for a given case. For details, see [Flowpath Parameterization Type in the TurboGrid User's Guide](#).

- Selecting the new **CAD Surfaces Are Watertight** option enables TurboGrid to make use of topological connectivity data stored in a CAD file, typically resulting in mesh generation that is faster and more reliable compared to when the option is not selected. For details, see [Complex Blade End Tab in the TurboGrid User's Guide](#).
- The **Suppress Inflation Layers on Named Selections** settings control the presence of inflation layers in the unstructured mesh block for a complex blade end. For details, see [Setting Descriptions for Complex Blade End Mesh Data Objects in the TurboGrid User's Guide](#).

- In the Mesh Data settings for a complex blade end, the new **Inflation Layer Mode** setting provides an option (Last Aspect Ratio) to limit the change in cell aspect ratio where the outermost inflation layer interfaces with the interior mesh. The setting's other option (Constant Expansion) gives the behavior of the previous release. For details, see [Setting Descriptions for Complex Blade End Mesh Data Objects in the TurboGrid User's Guide](#).
- The Global Size Factor (see [Method in the TurboGrid User's Guide](#)) affects the unstructured mesh blocks of complex blade ends closer to how it affects the passage mesh, in terms of element edge length and element count.

Ansys BladeModeler Release Notes

The following sections contain release information for Release 2025 R2 of Ansys BladeGen, Ansys BladeEditor, and Ansys BladeBuilder.

- 1. Supported Platforms
- 2. BladeGen
- 3. BladeEditor
- 4. BladeBuilder

1. Supported Platforms

Platform/OS levels that are supported in the current release are posted on the [Ansys website](#).

2. BladeGen

2.1. New Features and Enhancements

This section lists features and enhancements that are new in Release 2025 R2 of Ansys BladeGen.

Release 2025 R2 of BladeGen has no new features or enhancements.

3. BladeEditor

3.1. New Features and Enhancements

This section lists features and enhancements that are new in Release 2025 R2 of Ansys BladeEditor.

Release 2025 R2 of BladeEditor has no new features or enhancements.

4. BladeBuilder

4.1. New Features and Enhancements

This section lists features and enhancements that are new in Release 2025 R2 of Ansys BladeBuilder.

Release 2025 R2 of BladeBuilder has no new features or enhancements.

Polyflow Classic Release Notes

The following sections contain release information for Ansys Polyflow Classic 2025 R2.

1. [New Features](#)
2. [Supported Platforms](#)
3. [Resolved Issues and Limitations in Ansys Polyflow Classic 2025 R2](#)

1. New Features

The new features in Ansys Polyflow Classic 2025 R2 are as follows:

- Improved the usage of the MUMPS linear solver for distributed memory (clusters).
- Added Support for the CFD HPC Ultimate License
- Improved the performance of mesh adaption for shell simulation with contact

2. Supported Platforms

For information about past, present, and future operating system and platform support, see the [Platform Support section of the Ansys Website](#).

3. Resolved Issues and Limitations in Ansys Polyflow Classic 2025 R2

This section lists limitations that existed in previous releases, but that were removed in Ansys Polyflow Classic 2025 R2.

Forte Release Notes

The enhancements and defect corrections listed below are for Release 2025 R2.

1. [New Features and Enhancements](#)
2. [Resolved Issues and Limitations](#)

1. New Features and Enhancements

This section lists new features and enhancements in Release 2025 R2 of Ansys Forte CFD, organized by topic.

Engineering Models and Computation

- Revised the velocity direction bounding method at wall corners. This revision improves the accuracy of certain flow benchmark cases, but may cause the flow solution to be slightly different from previous releases.

2. Resolved Issues and Limitations

For a list of Ansys Forte issues that have been resolved in Release 2025 R2, refer to the [Resolved Issues and Limitations](#) document on the Ansys Help site.

Chemkin / Energico / Reaction Workbench / MFL

Release Notes

The following sections contain release information for Release 2025 R2 of Chemkin products:

1. [New Features and Enhancements](#)
2. [Resolved Issues and Limitations](#)

1. New Features and Enhancements

Chemkin

- New reactor models are now available for use with PyChemkin, including closed homogeneous batch reactor, plug flow reactor, steady-state perfectly-stirred reactor (PSR), PSR network with recycling streams, single-zone and multi-zone HCCI, SI internal combustion engine, premixed laminar flame-speed calculator, and premixed laminar burner-stabilized flame reactor models. New examples illustrating use of PyChemkin are available with the PSR reactor, network of PSR reactors, premixed flame-speed calculation, and premixed burner-stabilized flame reactor models. Documentation has been added for the PyChemkin user guide and API list in PDF format, and PyChemkin tutorials in PDF, Jupyter notebook, and html formats.
- Transport properties from the Real Gas Properties (RGP) table can now be used in Chemkin reactor models. Example usages are in the PSR, PFR, Engine-type models as well as in the coagulation kernel calculations.
- The licensing feature CFD HPC Ultimate is enabled for Chemkin. This allows running of parameter studies on multiple CPU cores in parallel without the need for additional HPC licenses.
- Robustness and usability improvement in Chemkin includes a more responsive user-interface.
- Added concentration and ppm variables to command-line options for post-processing through Get-Solution.

Reaction Workbench

- Improved performance for mechanism reduction by reducing time spent between running and post-processing operations. This will be especially noticeable when the mechanism being reduced is relatively small.
- Added an option to Mechanism Extraction to improve readability of the extracted mechanism file.
- Added explanation in the user interface that for property estimation, species can also be searched for with SMILES string in addition to atomic-composition based search.

Model Fuel Library (MFL)

- Now available: 33 new/improved Model Fuel Library (MFL) mechanisms to capture combustion chemistry of conventional and carbon-neutral fuels for transportation and power generation, and the chemistry of lithium-ion battery thermal runaway.

2. Resolved Issues and Limitations

For a list of issues that have been resolved in Release 2025 R2, refer to the [Resolved Issues and Limitations](#) document on the Ansys Help site.

FENSAP-ICE Release Notes

The following sections contain release information for Ansys FENSAP-ICE 2025 R2.

1. [New Features and Enhancements in Ansys FENSAP-ICE](#)
2. [Resolved Issues and Limitations](#)

1. New Features and Enhancements in Ansys FENSAP-ICE

• Ice Displacement and Multishot Remeshing Updates

Multishot remeshing workflows no longer require wall boundaries to be separated along sharp edges to properly capture these features during the wrapping process. This is a significant usability improvement, allowing the icing process to proceed without modifying the original grid boundary topology.

Non-conformal interfaces are removed from the mesh during remeshing. These boundaries are not required in the remeshed grid, as it is generated using unstructured elements. Additionally, the remeshing process cannot proceed if interfaces divide the domain into multiple volumetric regions.

Ice smoothing is automatically applied at wall/symmetry junctions in 3D remeshing simulations to improve the robustness of the surface wrapping process and reduce the likelihood of prism and tetrahedral meshing failures in these regions.

To improve robustness at wall/symmetry junctions, the wrap resolution factor has changed from 1 to 0.9 for 3D meshing cases.

The **STL ice thickness limit** option limits the iced STL file to iced facets only using a user-set ice thickness threshold. Limiting the STL to iced facets has the following benefits:

- STL file size is reduced, saving storage space.
- Excessive and spotty mesh refinements due to proximity detection between the STL and the original surface are avoided, resulting in smaller iced meshes during the multishot icing process.
- Remeshing time is noticeably reduced by reading a smaller STL file, minimizing proximity sizing checks, and using a more optimal sizing function during wrapping.
- More optimal iced grids with reduced mesh sizes allow larger problems with more detailed ice shapes to be solved using the same computing resources.
- STL files isolated to iced areas are easier to manipulate for downstream 3D printing processes, where the ice shapes can be manufactured and attached to actual aircraft surfaces.

A new restart option, called Reset FENSAP DROP, has been added to the Multishot configuration panel. This option skips shot-to-shot solution interpolation for FENSAP and DROP3D, avoiding memory issues associated with the `so ln2so ln` interpolation tool when very large grids are used. It enables the simulation to continue despite the large mesh size.

A new solution output mode, called **Write All Variables**, has been added to the FENSAP flow solver. When disabled, this option skips non-essential fields in the solution file, reducing its size and helping to avoid memory issues when DROP3D reads the file.

- **MPI, Solver, and Memory Upgrades**

FENSAP-ICE solver memory usage has been reduced by 35%, enabling larger problems to be solved using the same computational resources.

The default number of momentum linear system iterations in the FENSAP flow solver has been reduced from 10 to 5. This change significantly improves simulation speed without noticeably affecting convergence. Legacy runs configured with the previous default value of 10 will continue to use that setting unless changed manually. When a case is dragged and dropped into a new run, it will retain the original value of 10, which can be manually updated to 5. New runs will automatically be configured with the updated default value of 5.

A special set of Intel MPI flags is enabled by default to improve execution stability when running with a large number of cores and grids exceeding 100 million nodes.

When using the SLURM scheduler, the Fluent Meshing process is configured to distribute the mesh across HPC compute nodes rather than running entirely on the head node. This removes mesh size limitations during remeshing and helps prevent memory-related failures during tetrahedral meshing or while writing the newmesh.msh file. For other schedulers (for example, PBS), similar behavior can be achieved by replicating the SLURM configuration using system-specific options.

- **High Fidelity Multishot Ice Shape Simulations on Full-Scale Aircraft With Multiple Components**

With improvements to STL file management, the remeshing script, FENSAP memory usage, and MPI configuration, FENSAP-ICE can now compute detailed scallop ice shapes over entire wings of full-scale aircraft such as the NASA CRM-HL model. Simulations including additional components—such as the pylon, nacelle, and tail planes—are also supported. Multishot with remeshing runs have been tested using up to 1024 cores and mesh sizes reaching 450 million cells.

- **Constant Roughness on Iced Nodes Only**

A new roughness output mode has been added to ICE3D to generate a roughness.dat file with a user-defined, constant, and uniform roughness value applied only to iced nodes. This feature serves as an alternative to the beading model, allowing you to begin with smooth walls and apply a prescribed sand-grain roughness value in regions where ice is present.

- **Ice Shedding with Parallel Solver for Stress Evaluation (Beta)**

A new solver mode selection is available in the ice shedding model for rotating components. The legacy single-CPU solver is now labeled **Direct – Serial**, and the newly introduced parallel solver is labeled **Iterative – Parallel**. The parallel solver significantly improves performance, particularly for propeller ice shedding cases involving unstructured grids.

- **User Interface, User Experience, and Documentation Updates**

- Iteration history is now included in the **Minimal Log Output** mode, showing only equation residuals and solver time.
- ICE3D iteration history and crack propagation logs have been redesigned for improved readability.

- The FENSAP **Copy from Adiabatic** right-click menu has been removed from the FENSAP wall boundary temperature right-click menu to prevent configuration errors (for example, setting to Adiabatic + 10K).
- Sliding wall options have been relabeled in the ICE3D **Boundary Conditions** panel for clarity.
- The default ice view threshold field in Viewmerical has been changed from **Ice Growth (kg/m²)** to **Ice Thickness (m)**.
- The ICE3D numbered solution file format has changed from 8-digit to 6-digit numbering to align with Fluent Icing conventions.
- The y values in the FENSAP solution (soln) file are now limited to first-layer wall-adjacent cells only, as part of a memory optimization. This change removes the volume-to-surface cell projection data structure, which was unused elsewhere.
- [ICE3D - Ice Accretion and Water Runback](#) and [Multishot with Remeshing in the Ansys FENSAP-ICE User Manual](#) have been overhauled to include examples and guidelines aimed at clarifying various options and supporting successful multishot icing simulations.

2. Resolved Issues and Limitations

For a list of issues that have been resolved in Release 2025 R2, refer to the [Resolved Issues and Limitations](#) document on the [Ansys Help website](#).

EnSight Release Notes

The following sections contain release information for Ansys EnSight 2025 R2.

1. [New Features and Enhancements in Ansys EnSight](#)
2. [PyEnSight](#)
3. [Ansys Dynamic Reporting \(Nexus\)](#)
4. [Beta in This Release](#)
5. [Alpha in This Release](#)
6. [Resolved Issues and Limitations](#)

This document explains modifications and enhancements between EnSight releases. If you have not used EnSight before, you can safely skip the reading of this document.

Welcome to EnSight 2025 R2!

Help System

The Help system utilizes the Ansys Help Viewer. When working in online mode, the Ansys Help Viewer forwards help and documentation requests to the online location for help and documentation content. When working in offline mode, the Ansys Help Viewer redirects to a locally stored help bundle from Ansys. If you anticipate working offline, you should install the offline Help bundle from Ansys.

1. New Features and Enhancements in Ansys EnSight

The following list highlights the enhancements and changes to EnSight in 2025 R2. This list is very short due to significant reallocation of effort towards internal integration of our technologies into other tools. In addition, we have several major enhancements that have reached Beta state and can be utilized under the guidelines of a beta feature.

- **Improved DVS Exports from Fluent**

EnSight can now receive both user defined zones as well as volume and surface parts during a single export from Fluent. EnSight also receives report definitions and residuals information in the form of queries in EnSight. A collection of bugs/gaps in the DVS export from Fluent were fixed as well. As we move forward, we are striving to make the DVS data transport from Fluent to EnSight the suggested method.

- **Polyflow Users in Workbench**

Users can now utilize EnSight for post-processing Polyflow datasets within the Workbench environment. There is now an option when post-processing Polyflow datasets in Workbench to launch EnSight to view the results.

- **Ansys Results Reader Improvements**

We have resolved several outstanding bugs in the Ansys Results reader.

- **GLB Export Efficiency Improvements**

Transient with constant topology, when written to the single file format of GLB, now only writes a single array for the geometry (rather than 1 per timestep). This improvement only applies to single file GLB.

- **AVZ & JT Export with Vector Component Colors**

You can now correctly color their AVZ and JT Export with components of a vector directly. Previously, the color for vector coloration was only the magnitude of the vector. Now, if you select to color by a component of a vector, that color is correctly respected in the AVZ and JT Export.

- **Internal Library Updates**

Countless number of internal libraries have been updated (e.g. Qt, Python, FFMPEG, VTK, etc.) to maintain consistency with other applications and corporate wide requirements.

- **Removal of the XDMF2 and XDMF3 Readers from Installation Package**

These were removed due to incompatibility with a required update in our application binary interface. Should you require these readers, you will need to either continue with 2025 R1 or earlier packages, or provide updated ABI versions for us to include.

- **Save CSV Results from the Performance Tool**

The Performance Tool (In UserDefinedTools) no longer pushes results directly to the performance website. Rather, you can save the CSV form of that performance graph locally.

2. PyEnSight

The following list highlights the enhancements and changes to PyEnSight in 2025 R2.

- **TBD**

TBD

3. Ansys Dynamic Reporting (Nexus)

See the 2025 R2 version of the [Ansys Dynamic Reporting Release Notes](#) for enhancement and change information.

4. Beta in This Release

Documentation for the beta items can be found at [Ansys EnSight Beta Manual](#).

5. Alpha in This Release

We have several items that are marked as Alpha for this release. This means that the features have not yet been fully tested, documented, or have their intended functionality. They are meant to be used only upon the guidance of EnSight staff. These include:

- **DPF Reader**

Work continues on a new Mechanical results file reader based on the DPF library. Currently tested against Ansys Mechanical (.rst) file formats.

6. Resolved Issues and Limitations

For a list of issues that have been resolved in Release 2025 R2, refer to the [Resolved Issues and Limitations](#) document on the [Ansys Help website](#).

FreeFlow Release Notes

Ansys FreeFlow™Smoothed-Particle Hydrodynamics (SPH) Simulation Software

Ansys FreeFlow software was designed to provide an engineering tool to tackle complex free-surface flow applications, offering a detailed and realistic representation of physical processes that are challenging to simulate with other methods.

Ansys FreeFlow software offers a comprehensive platform to simulate single-phase free-surface flows, offering two different solver approaches, an implicit incompressible and a weakly compressible. Its powerful GPU-based solver manages large-scale simulations featuring millions of SPH elements efficiently.

- Single-phase SPH Solvers;
- Surface Tension Modeling;
- Thermal Solver;
- Coupling with Ansys Motion and Ansys EnSight;
- Air drag forces through Ansys Fluent coupling;
- Solver Customization and PrePost Scripting.

See more details about Ansys FreeFlow software on [Ansys Website](#)

Learn how-to use with:

- [Ansys FreeFlow User Manual](#)
- [Ansys FreeFlow Tutorial Guide](#)

1. New Features

In this section you will find all FreeFlow software new features present in the 2025 R2 version.

1.1. General New Features (SPH)

- New properties and curves for SPH such as element mass, viscosity, pressure gradients, and much more were created.

SPH Solver curves were created so users can have access to more information related to solver SPH iterations while SPH Properties were created to improve post processing custom expressions capabilities.

1.2. General New Features (Solver SDK)

- A support for custom heat transfer coefficient correlations based on local average velocity of the triangles was created.
- A support for custom variable for each SPH element was created
- A support for users to change gravity value was created.

1.3. Ansys FreeFlow Coupling with Ansys Mechanical Software

See below the new features you can use to perform the Ansys FreeFlow coupling with Ansys Mechanical™ software:

- **2-Way Structural Coupling (SPH-FEA) for Fluid-Structure Interaction (FSI):** FreeFlow software sends the SPH forces to Ansys Mechanical software, which calculates deformations and send the Displacements back to FreeFlow project.

2. Improvements

In this section you will find the enhancements done for this version of FreeFlow Software.

2.1. General Improvements

See below the general improvements that this new version brings:

- The support to rotate Volumetric Inlets was added.
- Transient Point clouds can now have custom filenames on each point cloud. The definition of what file belongs to which time is done in the main index file.
- Point clouds now support units for its quantities. The units are defined in the header of the point cloud files.
- Point clouds now support .csv and MS Excel files.
- Point clouds now support vector definitions on the header of each file.

For more details see [Ansys FreeFlow User Manual](#).

2.2. Performance Improvements

- Performance improvements on the search algorithm of point cloud were implemented.

2.3. Postprocessing Improvements

- The Cube and Cylinder groups which allow to a create a cube/cylinder over multiple entities (e.g. Geometries, SPH, Point Clouds) were added.
- Polyhedron user process can now be created on SPH.

- Coordinate as a property of Eulerian Statistics process was added.

For more details see [Ansys FreeFlow User Manual](#).

3. FreeFlow Modules

In this section you will find new FreeFlow Modules and main enhancements present in the 2025 R2 version.

- **Gravity Variable Module:** This module allows defining a time-dependent gravitational field. Users can specify the magnitude and direction of gravity acceleration as a function of time, enabling simulations in dynamic environments.
- **SPH Disable Wall Corrections Module:** The purpose of this module is to disable near boundary corrections implemented in the SPH calculations. In some cases, these corrections may introduce instabilities into the simulation. The module disables corrections for all walls in the simulation.
- **New Ready-to-use Modules Beta:**

Ansys, Inc. makes no commitment to resolve defects reported against these prototype features. However, your feedback will help us improve the overall quality of the product. We will not guarantee that the projects using this beta feature will run successfully when the feature is finally released so you may, therefore, need to modify the projects.

- **Temperature Dependent Viscosity Module (Beta):** The **Temperature Variable Viscosity Module** allows defining a temperature-dependent SPH viscosity. Users can specify the viscosity of the fluid as a function of temperature using a csv input file.

For more details see [Module Manual](#).

4. Beta in This Release

In this section you will find the new beta features available in this release. Note that beta features have not been fully tested and validated.

Ansys, Inc. makes no commitment to resolve defects reported against these prototype features. However, your feedback will help us improve the overall quality of the product. We will not guarantee that the projects using this beta feature will run successfully when the feature is finally released so you may, therefore, need to modify the projects.

Important:

To use a beta feature the **Experimental (Beta) Features** checkbox must be enabled on the Options | Preferences dialog.

- Performance improvements for visualization of high triangle count geometries.

For more details see [Ansys FreeFlow User Manual](#).

5. Resolved Issues and Limitations

For a list of issues that have been resolved in Release 2025 R2, refer to the [Resolved Issues and Limitations](#) document on the [Ansys Help website](#).

Ansys Rocky Release Notes

The following sections contain release information for Ansys Rocky™ particle dynamics simulation software 2025 R2.

This Rocky software release continues to address customers' needs delivering features that enable users to perform accurate DEM simulations, and optimize user experience, allowing limitless possibilities on particle dynamics simulation.

Welcome to **Ansys Rocky Software** 2025 R2!

1. New Features

In this section you will find all Rocky new features present in the 2025 R2 version.

1.1. General New Features (DEM)

- The support to import custom particles from .msh files was implemented.
- Bulk solid fraction is now a property of each material, instead of a global value.
- The integration between Coarse-Grain Modelling (CGM) and JKR adhesive force model was implemented.

For more details see [Rocky User Manual](#).

1.2. General New Features (SPH)

- New properties and curves for SPH such as element mass, viscosity, pressure gradients, and much more were created.

SPH Solver curves were created so users can have access to more information related to solver SPH iterations while SPH Properties were created to improve post processing custom expressions capabilities.

For more details see [Rocky User Manual](#).

1.3. General New Features (Rocky SDK)

- A new way to access Fluent reference pressure for CFD coupling modules was implemented.
- A support for custom heat transfer coefficient correlations based on local average velocity of the triangles was created.
- A support for custom variable for each SPH element was created

- A support for users to change gravity value was created.

For more details see [Rocky for Developers](#).

1.4. Ansys Rocky Coupling with Ansys Fluent Software

See below the new features you can use to perform the Ansys Rocky coupling with Ansys Fluent® software:

- **Semi-Resolved Thermal Support:** This feature adds support for Heat Exchange between Particle and Fluid in Semi-Resolved coupled Rocky-Fluent simulations. There are **two versions** of the implementation: one supporting **laminar flows** and another supporting **turbulent flows**
- For 2-way unresolved coupling, when running Fluent single phase, the porous media formulation was replaced by the DPM blocking formulation.
- The support for Fluent VOF on CFD semi-resolved coupling was added.

For more details see [Rocky User Manual](#), and [Rocky CFD Manual](#)

1.5. Ansys Rocky Coupling with Ansys Mechanical Software

See below the new features you can use to perform the Ansys Rocky coupling with Ansys Mechanical™ software:

- **2-Way Structural Coupling (SPH-FEA) for Fluid-Structure Interaction (FSI):** Rocky software sends the SPH forces to Ansys Mechanical software, which calculates deformations and send the Displacements back to Rocky project.

For more details see [Rocky User Manual](#).

1.6. Ansys Rocky Coupling with Ansys Motion Software

See below the new features you can use to perform the Ansys Rocky coupling with Ansys Motion™ software:

- Ansys Motion integration is now installed through Ansys automatic installer.

2. Improvements

In this section you will find the enhancements done for this version of Rocky Software.

2.1. General Improvements

See below the general improvements that this new version brings:

- The support to rotate Volumetric Inlets was added.
- Transient Point clouds can now have custom filenames on each point cloud. The definition of what file belongs to which time is done in the main index file.

- Point clouds now support units for its quantities. The units are defined in the header of the point cloud files.
- Point clouds now support .csv and MS Excel files.
- Point clouds now support vector definitions on the header of each file.

For more details see [Rocky User Manual](#).

2.2. Performance Improvements

- Performance improvements on the search algorithm of point cloud were implemented.
- The performance of diffusion equation mapping method do 2-way CFD-DEM unresolved coupling was improved.
- Performance improvements for loading properties through PrePost scripting.

2.3. User Experience Improvements

- CFD Coupling is now dynamic on the Data Tree, which means that only appear on the Data Tree if the simulation actually has CFD Coupling.

For more details see [Rocky User Manual](#).

2.4. Postprocessing Improvements

- The Cube and Cylinder groups which allow to a create a cube/cylinder over multiple entities (e.g. Geometries, SPH, Point Clouds) were added.
- Polyhedron user process can now be created on SPH.
- Coordinate as a property of Eulerian Statistics process was added.

For more details see [Rocky User Manual](#).

3. Rocky Modules

In this section you will find new Rocky Modules and main enhancements present in the 2025 R2 version.

New Ready-to-use Modules:

- **Edinburgh-Elasto-Plastic-Adhesion Module:** This module implements the Edinburgh Elasto-Plastic-Adhesive (EEPA) model in Rocky software and allows the tensile forces to develop, as well as a non-linear force-displacement behavior in compression.
- **DEM Liquid Spray Module:** This module enables you to model spray droplets using the DEM method. On top of that, the module gives users the ability to create a simple spray effect on spray droplets, spreading particles appart in a cone-like fashion, to represent a real spray pattern.

- **Gravity Variable Module:** This module allows defining a time-dependent gravitational field. Users can specify the magnitude and direction of gravity acceleration as a function of time, enabling simulations in dynamic environments.
- **SPH Disable Wall Corrections Module:** The purpose of this module is to disable near boundary corrections implemented in the SPH calculations. In some cases, these corrections may introduce instabilities into the simulation. The module disables corrections for all walls in the simulation.

- **New Ready-to-use Modules Beta:**

Ansys, Inc. makes no commitment to resolve defects reported against these prototype features. However, your feedback will help us improve the overall quality of the product. We will not guarantee that the projects using this beta feature will run successfully when the feature is finally released so you may, therefore, need to modify the projects.

- **Temperature Dependent Viscosity Module (Beta):** The **Temperature Variable Viscosity Module** allows defining a temperature-dependent SPH viscosity. Users can specify the viscosity of the fluid as a function of temperature using a csv input file.

For more details see [Rocky Module Manual](#).

4. Rocky Scripts

In this section you will find new Rocky Scripts in the 2025 R2 version.

New Ready-to-use Scripts:

- **Material Storage:** A ready-to-use script, for **particle properties management** (import, save, share, remove) using a local or Ansys Granta® databases.
 - **Material and Material interaction (particle-particle and particle-boundary, particle shape, size, physics and etc):**
 - Reuse calibrated materials;
 - Save new materials;
 - Fix incorrect or outdated entries;
 - Material data preview and editing;
 - Share Custom Data Base;
 - Supports custom STL (default scale: meters);
 - No support for flexible/breakable particles.

For more details see [Rocky Scripts Manual](#).

5. Beta in This Release

In this section you will find the new beta features available in this release. Note that beta features have not been fully tested and validated.

Ansys, Inc. makes no commitment to resolve defects reported against these prototype features. However, your feedback will help us improve the overall quality of the product. We will not guarantee that the projects using this beta feature will run successfully when the feature is finally released so you may, therefore, need to modify the projects.

Important:

To use a beta feature the **Experimental (Beta) Features** checkbox must be enabled on the Options | Preferences dialog.

- **Deformable Particles:** In this type of particle, the DEM elements that compose a flexible particle will actually deform, instead of only the joints. The deformable particles beta feature is useful for applications involving significant plastic deformation of particles, such as calendaring in battery manufacturing and tablet compaction in pharmaceuticals.

This feature enables particle large deformation through an FEM-inspired element deformation approach and it is compatible with discrete breakage model.

- Performance improvements for visualization of high triangle count geometries.
- See [Rocky Modules \(p. 157\)](#) section to review the Ready-to-use modules beta in this release.

For more details see [Rocky User Manual](#).

6. Resolved Issues and Limitations

For a list of issues that have been resolved in Release 2025 R2, refer to the [Resolved Issues and Limitations](#) document on the [Ansys Help website](#).

Release Notes: Thermal Desktop 2025 R2 Service Pack 2

This package contains service pack updates for Ansys products included in the unified installation package. Each service pack release for 2025 R2 will be listed separately, along with the related issues addressed, within this document.

Ansys Release:

2025 R2

Service Pack:

2025 R2.02

Release Date:

17/09/2025

Issues Addressed (by Product):

Thermal Desktop

2025 R2 2.1 (07/01/2026)

- The Thermal Desktop installation requires SpaceClaim version 2024 R2, which is no longer available for general download on the Ansys Customer Portal. Please contact your Ansys/Synopsys support representative if you do not already have access to SpaceClaim 2024 R2.

2025 R2 2.0 (17/09/2025)

- A bug was identified in 25R2 where conductors failed to generate between the first and last angular nodes at every subdivision of height. This problem was noticed in surfaces using centered nodes, and affected numerous surface types. This has been fixed.
- A bug was identified when using Check Initial State on the lump edit form. The results produced were obviously incorrect. This has been fixed.
- An issue was discovered where the current SAMG2024 LIBs and DLLs in the dependency repo were incorrect and were causing performance issues. This has been fixed.
- An issue was discovered where the **TdSaveAs** command was not working correctly in the 25R2 version. This has now been fixed.

2025 R2 1.0 (06/08/2025)

- An error has been corrected relating to the logic manager and array interpolation due to a unit conversion.
- Four new finite element types have been added to FE importer in Thermal Desktop:
 - BEAM 188
 - MPC184
 - SHELL181
 - SHELL281
- A bug in the FK locator has been fixed. This was with FK assignments in XREF files.

Ansys Electronics Products

Release notes are available for the following Ansys Electronics products:

[Icepak \(p. 165\)](#)

Icepak Release Notes

Release 2025 R2 of the Ansys Icepak application offers most of the capabilities from previous releases plus an enhancement.

- [Introduction \(p. 165\)](#)
- [New and Modified Features in Ansys Icepak 2025 R2 \(p. 165\)](#)
- [Resolved Issues and Limitations in Ansys Icepak 2025 R2 \(p. 165\)](#)

1. Introduction

Ansys Icepak 2025 R2 is a release of Ansys Icepak that has a new feature and resolved issues and limitations.

2. New and Modified Features in Ansys Icepak 2025 R2

Release 2025 R2 of Icepak has no new features or enhancements.

Ansys 24.2 will be the final release of the Classic Icepak platform. Classic Icepak will continue to be accessible for download beyond the 24.2 release from the Ansys Customer Portal. ACE support will formally continue for Classic Icepak until 31 December 2025.

3. Resolved Issues and Limitations in Ansys Icepak 2025 R2

For a list of issues that have been resolved in Release 2025 R2, refer to the [Resolved Issues and Limitations](#) document on the [Ansys Help website](#).

Ansys Optical Products

Release notes are available for the following Ansys optical products:

[Ansys SPEOS \(p. 169\)](#)

[SPEOS for NX \(p. 173\)](#)

[SPEOS for Creo Parametric \(p. 177\)](#)

[Ansys Optical Labs \(p. 181\)](#)

[SPEOS HPC \(p. 183\)](#)

Speos Release Notes

The following sections present the new features, enhancements and changes for Speos 2025 R2 release.

1. [New Features](#)
2. [Enhancements](#)
3. [Changes](#)
4. [Resolved Issues and Limitations](#)

1. New Features

This section describes the new features introduced in Speos 2025 R2 release.

MODTRAN Source/Sensor

- **BETA Feature:** As of version 2025 R2, you can create a Speos [MODTRAN source](#) using as input a radiance map file (*.tp5, *.json) generated out of the MODTRAN software. Therefore, you can simulate radiative transfer in the Earth's atmosphere by modeling the transmission of light through the atmosphere considering a complete spectrum from ultraviolet (UV) to far-infrared (FIR).



HPC: Ansys Cloud Burst Compute

- **BETA Feature:** [Ansys Cloud Burst Compute](#) is a new feature integrated into Ansys Speos, available as of version 2025 R2. It will allow you to seamlessly submit jobs on-demand to cloud equipped with the latest high-performance GPU/CPU hardware. The cloud resources are set up by Ansys and require no IT or HPC expertise to run large-scale simulations. Ansys Cloud Burst Compute supports CPU and GPU simulations.



glTF File Import

- As of version 2025 R2, Speos can [import glTF file formats](#). Thanks to glTF file formats, you can import and access assets (geometries + appearance properties) in Speos allowing you to create and simulate scenes with highly detailed models.



Ansyes Engineering Copilot

- As of version 2025 R2, the [Ansyes Engineering Copilot](#) feature is available in Speos. Activated from the Learning and Support tab, this feature eliminates the need to select options that take you to individual external web site resources. Instead, it provides a centralized location for the Learning, Forum, and Knowledge resources as well as direct access to Ansys training and the AnsysGPT™ virtual AI assistant.



2. Enhancements

This section describes the enhancements introduced in Ansys Speos 2025 R2 release.

Material

- AxF: Japanese characters are now better managed in AxF filenames when processed by Speos.

Live Preview

- As of version 2025 R2, new [Convergence](#) options are available in the Live Preview. Convergence options help you identify noisy areas in real-time during a preview of a Radiance simulation. As a consequence, you will be able to modify simulation parameters to minimize or eliminate the noise.



Simulation

- As of version 2025 R2, [GPU Inverse simulations](#) are now compatible with the Optimized Propagation option set to Relative. As a consequence, you benefit from the power of your GPU to rebalance XMP results and so remove potential noise.



- GPU and CPU Simulation algorithms have been aligned and improved to better manage **Fast Transmission Gathering** when a *.bsdf180 surface property file is applied on a body.

Optical Part Design: Optical Lens

- The TIR Lens source type is now compatible with the [Step Pillow beam type](#) when the **Orientation** type is set to **Inner** support.
- When enabling the [construction preview](#) for the Sources, you can now see the 3D meshing of TIR Lenses that can be built. A color code indicates the status: red for errors, orange for selection, and green for build-ready.



Component: Optical Design Exchange

- As of version 2025 R2, the [Optical Design Exchange](#) workflow has been improved to provide you with a better user experience.
 - The optical properties information is no longer integrated to the ODX feature. Now upon compute of an ODX feature, imported optical properties information are created as Material features in the Speos Simulation tree, and imported geometries are directly referenced in their respective Material features.
 - The meshing information are no longer integrated to the ODX feature. Now upon compute of an ODX feature, imported meshing information are created as a Local Meshing feature in the Speos Simulation tree, and imported geometries are directly referenced the Local Meshing feature.

As a consequence, the workflow has been standardized to correspond to a classical Speos workflow: indeed, you can now directly add to a Speos simulation the imported geometries from the Structure tree, as Materials and Local Meshing reference the geometries.



Virtual Reality Lab

- As of version 2025 R2, you can [export a selection of several views](#) to XMP files from the Measure > Advanced window.



3. Changes

This section describes the operational and support changes introduced in Ansys Speos 2025 R2 release.

Geometry Modeler

- As of version 2025 R2, *.scdoc file import (ACIS save format) is no longer supported. If you need to [convert a *.scdoc file](#) (ACIS save format) to a *.scdocx file (Parasolid save format), use the prior Speos version 2025 R1.

Simulation

- Inverse Simulation: The [Deterministic algorithm with Photon Map](#) (Build Photon Map, Load Photon Map, Build and Save Photon Map) is deprecated as of version 2025 R2 and will be in end of support in version 2026 R1.



Versions Update

- Windows: Windows 10 and Windows 11

Minimum NVIDIA Driver **572.60** is required from 2025 R2.

4. Resolved Issues and Limitations

For a list of issues that have been resolved in Release 2025 R2, refer to the [Resolved Issues and Limitations](#) document on the [Ansys Help website](#).

Speos for NX Release Notes

The following sections present the new features, enhancements and changes for Speos for NX 2025 R2 release.

1. New Features and Enhancements
2. Changes
3. Resolved Issues and Limitations

1. New Features and Enhancements

This section describes the new features and enhancements introduced in Speos for NX 2025 R2 release.

Optical Properties

- As of version 2025 R2, Speos for NX allows to use [AxF files](#) in Texture Mapping as optical properties. AxF files are textured appearance property files generated with X-Rites measurement devices.



HPC: Ansys Cloud Burst Compute

- **BETA Feature:** [Ansys Cloud Burst Compute](#) is a new feature integrated into Speos for NX, available as of version 2025 R2. It will allow you to seamlessly submit jobs on-demand to cloud equipped with the latest high-performance GPU/CPU hardware. The cloud resources are set up by Ansys and require no IT or HPC expertise to run large-scale simulations. Ansys Cloud Burst Compute supports CPU and GPU simulations.



Ansys Engineering Copilot

- As of version 2025 R2, the [Ansys Engineering Copilot](#) feature is available in Speos for NX. Activated from the Learning and Support tab, this feature eliminates the need to select options that take you to individual external web site resources. Instead, it provides a centralized location for the Learning, Forum, and Knowledge resources as well as direct access to Ansys training and the AnsysGPT™ virtual AI assistant.



Sensor

- As of version 2025 R2, a new layer separation is available: **Data Separated by Group of Faces**. Now you can create several groups of faces. Each group will then correspond to one layer in the result. Faces not included in a group will be added to the layer named **All layers** in the result. Thus, you will generate lighter results, and so be able to assess the contribution of mechanical parts on stray light. This applies to [Irradiance](#), [Intensity](#), [Radiance Frame](#), [Radiance Camera](#), [Radiance Observer](#), [3D Energy Density](#) sensors.



Live Preview

- As of version 2025 R2, new [Convergence](#) options are available in the Live Preview. Convergence options help you identify noisy areas in real-time during a preview of a Radiance simulation. As a consequence, you will be able to modify simulation parameters to minimize or eliminate the noise.



Simulation

- As of version 2025 R2, [GPU Inverse simulations](#) are now compatible with the Optimized Propagation option set to Relative. As a consequence, you benefit from the power of your GPU to rebalance XMP results and so remove potential noise.



- GPU and CPU Simulation algorithms have been aligned and improved to better manage **Fast Transmission Gathering** when a *.bsdf180 surface property file is applied on a body.

Virtual Reality Lab

- As of version 2025 R2, you can [export a selection of several views](#) to XMP files from the Measure > Advanced window.



2. Changes

This section describes the operational and support changes introduced in Ansys Speos for NX 2025 R2 release.

Versions Update

- Windows: Windows 10 and Windows 11

Minimum NVIDIA Driver **572.60** is required from 2025 R2.

Operating System

- NX 2212 is no longer supported as of 2025 R2.
- NX 2306 is no longer supported as of 2025 R2.
- NX 2312 up to NX 2312 (2312.9140) service pack is now supported as of 2025 R2.
- NX 2406 up to NX 2406 (2406.8500) service pack is now supported as of 2025 R2.
- NX 2412 is no longer in BETA and now officially released as of 2025 R2.

NX 2412 up to NX 2412 (2412.4001) service pack is now supported as of 2025 R2.

- NX 2506 is in BETA mode as of 2025 R2.

Deprecated and End of Life Features

- The [Deterministic algorithm with Photon Map](#) (Build Photon Map, Load Photon Map, Build and Save Photon Map) is deprecated as of version 2025 R2 and will be in end of support in version 2026 R1.



- The [Camera Post Processing](#) is deprecated as of version 2025 R2 and will be in end of support in version 2026 R1.

3. Resolved Issues and Limitations

For a list of issues that have been resolved in Release 2025 R2, refer to the [Resolved Issues and Limitations](#) document on the [Ansys Help website](#).

Speos for Creo Parametric Release Notes

The following sections present the new features, enhancements and changes for Speos for Creo Parametric 2025 R2 release.

1. [New Features and Enhancements](#)
2. [Changes](#)
3. [Resolved Issues and Limitations](#)

1. New Features and Enhancements

This section describes the new features and enhancements introduced in Speos for Creo Parametric 2025 R2 release.

Live Preview

- The [Live Preview](#) is now available in Speos for Creo Parametric. This allows you to compute your Direct and Inverse simulations using progressive rendering with the most powerful GPU available on your computer, and displays the result in a dedicated window calculated in real-time.



GPU

- You can now run your simulations using the available GPUs of your computer directly into Speos for Creo Parametric, and so without having to open Speos Core.

The [GPU Regenerate](#) option runs simulations on your computer using the cores of your GPU. The GPU simulation offers a faster alternative than classic CPU simulation.



HPC: Ansys Cloud Burst Compute

- **BETA Feature:** [Ansys Cloud Burst Compute](#) is a new feature integrated into Speos for Creo Parametric, available as of version 2025 R2. It will allow you to seamlessly submit jobs on-demand to cloud equipped with the latest high-performance GPU/CPU hardware. The cloud resources are set up by Ansys and require no IT or HPC expertise to run large-scale simulations. Ansys Cloud Burst Compute supports CPU and GPU simulations.



Ansys Engineering Copilot

- As of version 2025 R2, the [Ansys Engineering Copilot](#) feature is available in Speos for Creo Parametric. Activated from the Learning and Support tab, this feature eliminates the need to select options that take you to individual external web site resources. Instead, it provides a centralized location for the Learning, Forum, and Knowledge resources as well as direct access to Ansys training and the Ansys-GPT™ virtual AI assistant.



Camera Sensor

- UI improvement on the [Camera sensor](#). The definition of the Camera sensor now adapts to the distortion model used as input.



Simulation

- GPU and CPU Simulation algorithms have been aligned and improved to better manage **Fast Transmission Gathering** when a *.bsdf180 surface property file is applied on a body.

Virtual Reality Lab

- As of version 2025 R2, you can [export a selection of several views](#) to XMP files from the Measure > Advanced window.



2. Changes

This section describes the operational and support changes introduced in Ansys SPEOS for Creo Parametric 2025 R2 release.

Versions Update

- Windows: Windows 10 and Windows 11

Minimum NVIDIA Driver **572.60** is required from 2025 R2.

Operating System

- Version of Creo 7 is no longer supported as of 2025 R2.
- Versions of Creo 8 from 8.0.3.0 to 8.0.12.0 are now supported as of 2025 R2.
- Versions of Creo 9 from 9.0.1.0 to 9.0.10.0 are now supported as of 2025 R2.

- Versions of Creo 10 from 10.0.0.0 to 10.0.7.0 are now supported as of 2025 R2.
- Versions of Creo 11 from 11.0.0.0 to 11.0.3.0 are now supported as of 2025 R2.

Deprecated and End of Life Features

- The [Deterministic algorithm with Photon Map](#) (Build Photon Map, Load Photon Map, Build and Save Photon Map) is deprecated as of version 2025 R2 and will be in end of support in version 2026 R1.



- The [Camera Post Processing](#) is deprecated as of version 2025 R2 and will be in end of support in version 2026 R1.

3. Resolved Issues and Limitations

For a list of issues that have been resolved in Release 2025 R2, refer to the [Resolved Issues and Limitations](#) document on the [Ansys Help website](#).

Speos Labs Release Notes

The following sections present the new features and changes for Speos Labs 2025 R2 release.

- 1. [New Features and Enhancements](#)
- 2. [Changes](#)
- 3. [Resolved Issues and Limitations](#)

1. New Features and Enhancements

This section describes the new features and enhancements introduced in Speos Labs 2025 R2 release.

Virtual Reality Lab

- As of version 2025 R2, you can [export a selection of several views](#) to XMP files from the Measure > Advanced window.



2. Changes

This section describes the operational and support changes introduced in Ansys Optical Labs 2025 R2 release.

- Windows: Windows 10 and Windows 11

Minimum NVIDIA Driver **572.60** is required from 2025 R2.

3. Resolved Issues and Limitations

For a list of issues that have been resolved in Release 2025 R2, refer to the [Resolved Issues and Limitations](#) document on the [Ansys Help website](#).

Speos HPC Release Notes

The following sections present the changes and enhancements for Speos HPC 2025 R2 release.

[1. New Features and Enhancements](#)

[2. Resolved Issues and Limitations](#)

1. New Features and Enhancements

GPU

- Windows: Windows 10 and Windows 11

Minimum NVIDIA Driver **572.60** is required from 2025 R2.

- Linux

Minimum NVIDIA Driver **570.124.04** is required from 2025 R2.

Simulation

- Speos HPC has been improved and now better manages GPU simulations on Linux clusters.

2. Resolved Issues and Limitations

For a list of issues that have been resolved in Release 2025 R2, refer to the [Resolved Issues and Limitations](#) document on the [Ansys Help website](#).

Ansys Acoustics Products

Release notes are available for the following Ansys Acoustics Products:

[ANSYS Sound \(p. 187\)](#)

Ansys Sound Release Notes

The following sections present the new features, enhancements and changes for Ansys Sound™ products in the 2025 R2 release.

1. [Ansys Sound: Analysis and Specification](#)
2. [Ansys Sound: ASDforEV](#)
3. [Ansys Sound: Jury Listening Test](#)

1. Ansys Sound: Analysis and Specification

The following sections present the new features and enhancements for the Ansys Sound: Analysis and Specification™ application 2025 R2 release.

1.1. New Features

ISO 532-2 Loudness Indicator

Support has been added for the ISO 532-2 [loudness](#) psychoacoustic indicator.

ISO/TS 20065 Tonal Audibility Indicator

Support has been added for the ISO/TS 20065 [tonal audibility](#) psychoacoustic indicator.

PyAnsys Sound

Support for the Sound Composer application and multiple psychoacoustic indicators has been added to [PyAnsys Sound](#).

1.2. Enhancements

Psychoacoustic Indicator Results Table

The [results table](#) for psychoacoustic indicators has been refined and now includes a feature to display multiple selected curves in the same graph.

Selecting Part of a Temporal Signal

The mechanism for [selecting part of a temporal signal](#) between two cursors has been improved to allow selection from either the toolbar, menu, or using a right-click.

1.3. Resolved Issues and Limitations

For a list of issues that have been resolved in Release 2025 R2, refer to the [Resolved Issues and Limitations](#) document on the [Ansys Help website](#).

2. Ansys Sound: ASDforEV

The following sections present the new features and enhancements for the Ansys Sound: ASDforEV™ application 2025 R2 release.

2.1. Enhancements

UI Refinements

The ASDforEV user-interface has been updated for improved usability.

2.2. Resolved Issues and Limitations

For a list of issues that have been resolved in Release 2025 R2, refer to the [Resolved Issues and Limitations](#) document on the [Ansys Help website](#).

3. Ansys Sound: Jury Listening Test

The following sections present the new features and enhancements for the Ansys Sound: Jury Listening Test™ application 2025 R2 release.

3.1. Resolved Issues and Limitations

For a list of issues that have been resolved in Release 2025 R2, refer to the [Resolved Issues and Limitations](#) document on the [Ansys Help website](#).

Ansys Autonomous Vehicle Products

Release notes are available for the following Ansys Autonomous Vehicle products:

[Ansys AVxcelerate Lighting and Sensors \(p. 191\)](#)

[Ansys AVxcelerate Sensors \(p. 193\)](#)

AVxcelerate Lighting and Sensors Release Notes

The following section presents the enhancements and changes for AVxcelerate Lighting and Sensors 2025 R2 release.

[1. Enhancements](#)

[2. Change](#)

1. Enhancements

This section describes the enhancements introduced in AVxcelerate Lighting and Sensors 2025 R2 release.

2. Change

This section describes the change introduced in AVxcelerate Lighting and Sensors 2025 R2 release.

AVxcelerate Sensors Release Notes

The following sections present the new features, enhancements and changes for the AVxcelerate Sensors™ solution 2025 R2 release.

- 1. New Features
- 2. Enhancements
- 3. Changes

1. New Features

This section describes the new features introduced in AVxcelerate Sensors Simulator 2025 R2 release.

1.1. New Rendering Engine for Camera Simulations

The new Light Propagation Engine, also referred to as LPE, released as a beta feature, is a powerful rendering engine that you can now use for camera simulations instead of the default rendering engine.

The Light Propagation Engine not only offers a better and more realistic rendering, it also increases the simulation speed.

If you want to use the Light Propagation Engine for camera simulations, you need the **AVxcelerate Sensors Library for LPE**, which provides you with resources (assets and tracks) specifically prepared for this rendering engine. To do so, you must select this option when installing the AVxcelerate Sensors solution.

You will then have to configure the **LPE Parameters** in the [camera simulation parameters](#). You will find three examples of simulation parameters files in the dedicated library.

1.2. Direct Data Memory Access

The DDMA feature, which stands for Direct Data Memory Access, is a new method to get the output data produced by sensors.

Note:

In this release, DDMA is delivered as a beta feature available on Windows only, and it supports output data of physics-based camera sensors only.

Particularly suited for HiL and SiL use cases, DDMA offers better performance compared to data access from the shared memory since it provides direct access to raw data in CPU.

Thanks to the DDMA Software Development Kit (SDK) delivered in the product, you can develop your own dynamic link library (dll) customized to your needs.

The RDMA SDK is composed of:

- the CPU DDMA sample, which demonstrate how to build a custom dll that will be used to retrieve camera output data from the CPU,
- the *ProcessSensorData.h* header file, which defines the API methods that should be implemented in your code,
- the DDMA extension, which enables AVxcelerate Sensors Simulator to dynamically load and execute a custom dll during a simulation.

For more information, refer to the readme file provided in the sample folder or to the [Raw Data Output for DDMA](#) page of the User's Guide.

1.3. Radar Phase Noise

In order to perform a radar simulation in a context which more closely resembles real word scenarios, in the 2025 R2 release of AVxcelerate Sensors Simulator, it is now possible to simulate phase noise in radar modes with a System FMCW or Performance FMCW waveform. To create a radar model with phase noise, use the Sensor Labs application, or the Sensors REST API to model phase noise in one of three ways:

- using custom values for the Pedestal Model (Pedestal Parametric Model)
- using Preset values based on the Pedestal Model
- using the Piecewise Linear Model.

To simulate phase noise, simply add your radar to a sensor layout, configure the radar simulation parameters, and run the simulation. For more information on this parameter, refer to the [Mode Configuration Parameters](#).

1.4. Radar Range Filters

To counteract the aliasing phenomenon and improve the signal-to-noise ratio (SNR), you can now add range filters to your sensor model using the Sensor Labs application or the Sensors REST API. Set at the mode level, you can configure up to two range filters per mode in a radar model. You can use one of three ways to define attenuation and create:

- low-pass filters to suppress low-frequency components,
- high-pass filters to suppress high-frequency components,
- both a low-pass and a high-pass filter to mimic the behavior of a bandpass filter.

For more information, refer to [Range Filter](#) or to the [Mode Configuration Parameters](#) in the AVxcelerate Sensors Simulator User's Guide.

1.5. Simulation Reset

In the 2025 R2 release of AVxcelerate Sensors Simulator, you can optimize performance when executing consecutive simulation runs thanks to the ability to perform a simulation reset. Using the [simulation](#)

[control procedures](#), you can now call an **Initialize** command directly after a **Stop** command. This triggers the resetting of the scenario using the resources that were loaded from the initial simulation for each subsequent simulation run, thereby optimizing on load time. In the current release:

- only sensors and advanced lighting systems are automatically reset.
- any changes to the asset or track during the previous simulation run must be manually reset to the desired initial settings through a world update during the **Initialize** command. This includes for example, changes in asset positions, orientations, states, date/time.
- noise is not reset.

When a simulation reset is performed and depending on your data access settings, the output files are organized accordingly. Refer to the [Results](#) section of the AVxcelerate Sensors Simulator User's Guide for more information.

Note:

Simulation reset is a beta feature in the 2025 R2 release. For more information, see [Simulation States and Transitions](#) in the AVxcelerate Sensors Simulator User's Guide.

2. Enhancements

This section describes the enhancements introduced in AVxcelerate Sensors Simulator 2025 R2 release.

2.1. Remote Data Memory Access

Leveraging National Instruments RDMA technology, the RDMA feature of AVxcelerate Sensors Simulator allows you to transfer raw data generated by physics-based camera sensors to an RDMA receiver.

Note:

In this release, RDMA is delivered as a beta feature available on Windows only.

In the previous release, you could transfer camera data via RDMA only by using the pre-built RDMA sender available on demand.

In the 2025 R2 release, you now have two options to [transfer raw data via RDMA](#):

- [using the pre-built RDMA sender](#), which is still provided upon request to Ansys Support,
- [using a custom dll](#), which should be built based on:
 - the NI RDMA SDK,
 - the DDMA SDK,
 - the resources provided in the CPU RDMA sample.

Using a custom dll has the many advantages over using the pre-built RDMA sender.

First, when developing a custom dll, you can add any data post-processing or conversion steps you need, for example to convert RGBA data to aRGB, or to turn the spectral irradiance map data produced by a camera with the **Lens Output (Light)** output type into an RDMA compatible format.

Also, the sensor identifier being provided as a metadata, along with all the information needed to decode the data, it is possible to get the output data of several camera sensors. When using the pre-built RDMA sender, the sensor identifier is not part of the metadata, data identification is based on the timestamp only, which is error-prone when retrieving the output data of several sensors.

Finally, the connection between the receiver and the sender can be established faster.

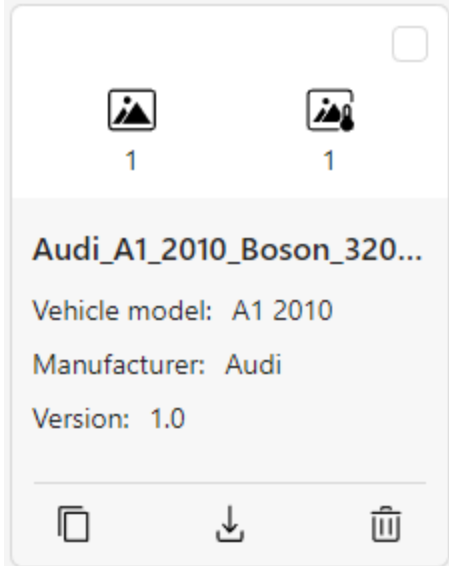
2.2. Thermal Camera Sensor Workflow Enhanced

In order to ease the configuration of sensor layouts including thermal camera models, the Sensor Labs applications and REST APIs now partially support thermal cameras.

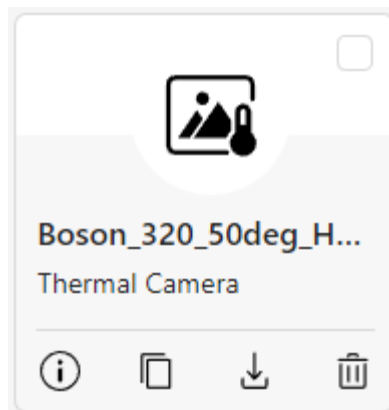
You can now:

- upload, edit and download sensor layout files (*.sencfgx) that contain thermal camera sensors using the Sensor Layout application or Sensor Layouts REST API v2,
- see the thermal camera sensors in your Sensors Library and download them as *.thermalpbcamx files using the Sensor Modeler application or Sensors REST API v7,

Layout containing a thermal camera



Thermal camera in Sensors Library



Important notes: The thermal camera sensor models cannot be opened or edited using the Sensor Modeler application, nor be created, updated or read using Sensors REST API v7.

The simulation of a thermal camera sensor now produces [temperature maps](#) only, available as text files or as serialized byte arrays in the shared memory. Grayscale images in png format are no longer generated.

This is a breaking change that requires you to update the [simulation parameters of your thermal camera sensor](#) so that the camera recording format must be set to **TEMPERATURE_MAP** instead of **PNG** if you want to save the thermal camera output files to disk.

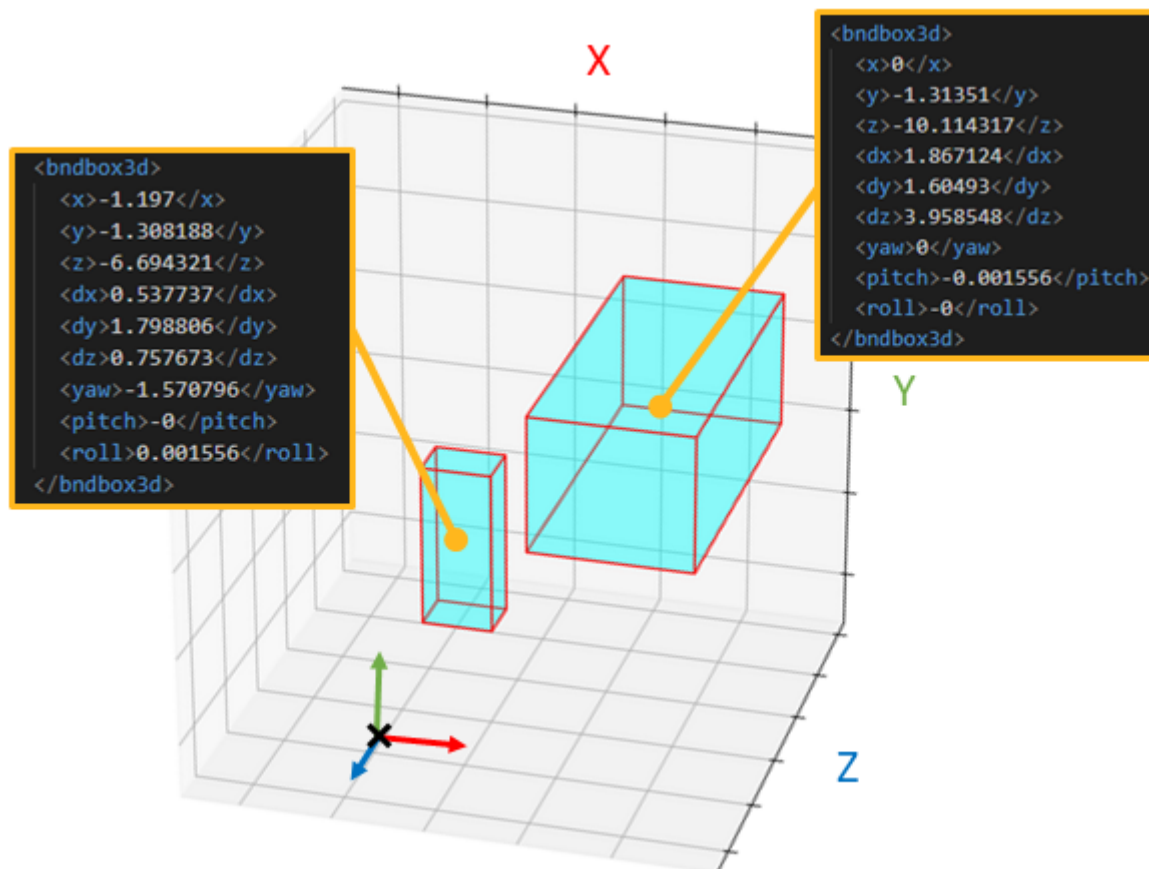
2.3.3D Bounding Box

3D bounding boxes are now part of the ground truth data that camera sensors can generate.

The 3D bounding box data are generated along with the 2D bounding box data when the **generate2dBoundingBoxes** camera simulation parameter is set to **true**. The *2dboundingBoxes.xml* files now includes both the 2D and 3D bounding boxes.

While a 2D bounding box is defined with pixel coordinates in the image frame, a 3D bounding box is defined by:

- the position of its isobarycenter specified with the **x,y** and **z** coordinates in the camera reference frame, expressed in meters,
- its dimensions, with **dy** being the height, **dx** the width and **dz** the depth of the bounding box, expressed in world meters,
- its orientation relative to the camera's axis system as a **yaw - pitch - roll** sequence, expressed in radians.



For more details, refer to the **Bounding Box** section of the [Camera Ground Truth Output Data](#) page.

Note:

In this release, 3D Bounding box is delivered as a beta feature.

2.4. Simulation Improvements

The following sensors simulation improvements have been made in the AVxcelerate Sensors 2025 R2 release:

- It is now possible to optimize simulation performance by adding the new **-s | --singleExecutable** argument to run a single-sensor simulation using only the `avxcelerate.sensorssimulator.exe`. Refer to [Simulation Process Arguments](#) for more information on using this simulation argument.
- Surface curvature can now be taken into account in the time of flight calculation for LiDAR simulation thanks to the addition of the **enableSurfaceSmoothing** parameter in the [LiDAR Simulation Parameters](#).

3. Changes

This section describes the changes introduced in AVxcelerate Sensors 2025 R2 release.

3.1. Supported Operating Systems

Ubuntu 24.04 is now a Linux distribution compatible with AVxcelerate Sensors Simulator. Ubuntu 20.04 is no longer supported. For the complete list of compatible operating systems and distributions, refer to the Platform Support by Application / Product document published [here](#) on the Ansys Website.

3.2. Compatibility with driving simulators

3.2.1. Co-simulation with SCANeR 2023.2

AVxcelerate Sensors Simulator is now also compatible with SCANeRstudio™ version 2023.2.

On Windows, you can now install the AVxcelerate Sensors Library for SCANeR either for SCANeRstudio™ version 2021.1 or for SCANeRstudio™ version 2023.2.

On Linux however, SCANeRstudio™ is no longer supported.

3.2.2. CarMaker 11.0 and 11.1 no longer supported

AVxcelerate Sensors Simulator is no longer compatible with CarMaker 11.0 and 11.1 versions. The AVxcelerate Sensors Library for CarMaker no longer provides resources for co-simulation with CarMaker 11.

3.3. Folder and file names changed

When installing the AVxcelerate Sensors solution, its main software components are no longer installed in *Optical Products/VRXPERIENCE* on Windows and *OpticalProducts/VRXPERIENCE* on Linux, the subfolders are now named *Autonomy/AVxcelerateSensors*. The same changes have been made to the folders in %appdata%, %localappdata% and %programdata%.

The below-listed executable, log and configuration files have also been renamed.

2025 R1	2025 R2
VRXPERIENCE.SensorsSimulator.exe	avxcelerate.sensorssimulator.exe
Imagine.Pre.VssSpawner.exe	avxcelerate.sensorprocess.exe
VRXPERIENCE_Sensors_dataAccessServer.exe	avxcelerate.sensorsdataaccessserver.exe
VRXPERIENCE_SENSORS.exe	avxcelerate.scanerconnector.exe
VRXPERIENCE.SensorsSimulator.log	avxcelerate.sensorssimulator.log
Imagine.Pre.VssSpawner[...].log	avxcelerate.sensorprocess[...].log
VRXPERIENCE.SensorsSimulator.dll.config	avxcelerate.sensorssimulator.dll.cong
Imagine.Pre.VssSpawner.dll.config	avxcelerate.sensorprocess.dll.config

Ansys Mesh Prep Products

Release notes are available for the following Ansys Mesh Prep products:

[Meshing \(p. 203\)](#)

[ICEM CFD \(p. 209\)](#)

Meshing Application Release Notes

Release 2025 R2 of the Meshing application contains many new features and enhancements. Areas where you will find changes and new capabilities include the following:

1. [Changes in Product Behavior from Previous Releases](#)
2. [Enhancements to Mesh Workflows](#)
3. [Enhancements to MultiZone Method](#)
4. [Enhancements to Pull](#)
5. [Enhancements to Weld](#)
6. [Enhancements to Mesh Details View](#)
7. [Enhancements to Local Sizing Control Options](#)

Many of the enhancements detailed in the [Mechanical Application Release Notes](#) are also relevant to the Meshing application.

1. Changes in Product Behavior from Previous Releases

The **2025 R2** release includes the following changes in product behavior:

- Element Normals are the same across the meshing methods and Ansys releases. However, orientation of elements such as Elemental Triads, Nodal Triads, and Elemental XY/YZ directions may vary.
- In Mesh Workflows,
 - → **Create Topology** operation is renamed to **Build Topology**.
 - Some options are renamed as follows in **Hemispherical Enclosure**:
 - **Center Modality** to **Center Type**
 - **Orientation Modality** to **Orientation Type**
 - **Open Closure** to **Open Enclosure**
 - **Orientation Modality** is renamed to **Orientation Type** in **HemiConvex Irregular Shape Enclosure** control.
 - **Checkpoint** control cannot be added from a Step using Insert option. You should select **Create Checkpoint** as **Yes** in the respective **Steps Details** view of the operation to create checkpoints.
 - You can now use **Control Type** dropdown to select the available control types.

- In **MultiZone Method**, the **Surface Mesh Method, PrimeMesh Quad Dominant** is renamed to **PrimeMesh**.
- In **Automatic Methods** under **Mesh Details** view, the **Sheet Body Method, PrimeMesh** is renamed to **Prime Quad Dominant**.

2. Enhancements to Mesh Workflows

Mesh Workflows has been enhanced to:

- Support Stacker workflow

Stacker Workflow maps a 3D model to a 2D plane and generates a 2D mesh to virtually stack a 3D mesh representing the input model. The workflow contains steps to:

- Detect the stackable bodies
- Stack the bodies to 2D plane
- Generates a 2D mesh to virtually stack a 3D mesh
- Delete the base face (2D Mesh)

For more detailed information, see [Stacker Mesh Workflow](#).

- Provide buttons for **Mesh Edge Tessellation** and **Geometry Edge Tessellation** in the **Domain Browser**.
- Provide the following new operations:
 - **Create Labels** to create labels for the node, edge, face and volume based on the scoped entities or proximity radius
 - **Prescribed Points** to add a set of prescribed points and associated labels to a volume mesh which can be propagated as Vertices with Named Selections at the **Output**.
 - **Merge Parts** to merge all parts defined in the scope to a single part.

Note:

Merge Parts may be required for specific meshing procedures like **Connect Topology** or **Merge Nodes** to provide conformal mesh across the volumes within a part.

- **Merge Nodes** to merge the source nodes to target nodes within the specified merge node tolerance
- **Create Zones** to create zones from the scoped parts, zones or labels.
- **Model Diagnostics** to diagnose the topology and surface mesh issues
- **Repair Topology** to repair topologies of the scoped entities

- **Auto Repair Topology** to automatically repair topologies of the scoped entities
 - **Replicate Mesh** to duplicates the volume mesh pattern onto another location in the model
 - **Improve Surface Mesh** to improve the surface mesh generated in the **Mesh Surface** step.
- Provide new controls and outcomes in **Mesh Surface** operation.

The new controls are as follows:

- **Quad Layer**
- **Geometry Fidelity**
- **Surface Mesh Diagnostics User Name**
- **Topology Diagnostics User Name**
- **Quad Mesh Advanced Options**

The new **Outcomes** are as follows:

- **Topology Diagnostics**
 - **Surface Mesh Diagnostics**
 - **Quality Metrics**
- Provide new options in **Size Field Surface Mesher** and **Constant Size Surface Mesher** as follows:
 - **Project On Geometry**
 - **Retain Existing Mesh**
 - Provide a new option **Use CAD Curvature** in **Curvature Sizing**.
 - Provide a new option **Project Side Faces** in **Extrusion** control. **Project Side Faces** is useful for an NVH case defined with a symmetry plane particularly when you use a hemi-convex enclosure and you want the side faces to be on the symmetry plane after extrusion.
 - Provide new options under **Output** object. They are:
 - **By Topology** in **Data Transfer Type**
 - **All Zones** in **Process Entities of Type**
 - **Transfer Labels to Named Selections**
 - **Transfer Prescribed Points Labels To Named Selections**
 - **Label Exclude Pattern**
 - **Label Include Pattern**
 - Provide a new option **Input Data** to define the type of input in the **Input** object.

- **Property Worksheet** has been enhanced to provide:
 - **Description** column that you can edit to provide a description.
 - **Show Association** checkbox to display **Association** column in the **Property Worksheet** when you select the checkbox.
 - Right-click options in the **Property Worksheet**:
 - **Define By Value**
 - **Define By Settings**

For more detailed information, see [Mesh Workflows](#).

3. Enhancements to MultiZone Method

MultiZone Method has been enhanced to:

- Rename the **PrimeMesh Quad Dominant** to **PrimeMesh** in **Surface Mesh Method**.
- Rename **Medial Axis** to **Axis Sweep** in the **Decomposition Type**.
- Improve **Axis Sweep** in **Decomposition Type** to give better mesh distribution when the sweep path bends around complex corners.

For more detailed information, see [MultiZone Mesh Method](#).

4. Enhancements to Pull

Pull has been enhanced to provide a new option **Auto Rename Bodies** to name the generated pull bodies with the original name of the sheet bodies from which it is generated. The **Auto Rename Bodies** is available only when **Associate To Pull Geometry** is **Yes** in the **Mesh Edit** Details view.

For more detailed information, see [Pull](#).

5. Enhancements to Weld

Weld has been enhanced to provide:

- **Weld Detection Method**
- Increased **Number of HAZ** layers to 6.

Weld Worksheet has been enhanced to provide:

- **Go To Generated Weld Bodies in Tree**
- **Go To Construction Weld Curves in Tree**

For more detailed information, see [Weld](#).

6. Enhancements to Mesh Details View

Mesh Details view has been enhanced to include the following:

- **Advanced Group** has been enhanced to provide **Auto-Map Fillets**. **Auto-Map Fillets** detect and generate map mesh on fillets using the defined fillet radius to enhance curvature capture on fillets with coarse mesh resolution when **Auto-Map Fillets** is **Yes**.

For more detailed information, see [Auto-Map Fillets](#).

- In **Automatic Methods** under **Mesh Details** view, the **Sheet Body Method, PrimeMesh** is renamed to **Prime Quad Dominant**. When the **Sheet Body Method** is **Prime Quad Dominant** and fails to generate mesh on bodies with controls that are incompatible with **Automatic(PrimeMesh)** method, the mesher automatically meshes those bodies using Quad Dominant method.

For more detailed information, see [Automatic Methods](#).

7. Enhancements to Local Sizing Control Options

Local Sizing Control Options has been enhanced to have **Blend to Neighbors** option to refine the neighbouring edges based on the applied edge sizing control on the selected edge when **Blend to Neighbors** is **Yes**. The default value is **Yes**. When **Blend to Neighbors** is **No**, the mesher does not refine the neighbouring edges of the edge with the applied edge sizing control.

For more detailed information, see [Descriptions of Local Sizing Control Options](#).

ICEM CFD

Release 2025 R2 development efforts included enhancement of Ansys ICEM CFD as a standalone application as well as continued development of its underlying technology exposed within the Ansys Workbench-based Meshing application.

Ansys ICEM CFD 2025 R2 has no new features or enhancements.

Ansys Connect Products

Release notes are available for the following Ansys Connect products:

[Ansys Minerva \(p. 213\)](#)

[Distributed Compute Gateway \(p. 219\)](#)

[optiSLang \(p. 221\)](#)

Ansys Minerva

The following topics provide an overview of what is new and improved in Ansys Minerva 2025 R2:

- 1. New Features and Enhancements
- 2. Resolved Issues
- 3. Known Issues and Limitations
- 4. Documentation Enhancements

1. New Features and Enhancements

New features and enhancements are available in the following areas of Ansys Minerva 2025 R2:

- 1.1. Installation and Setup
- 1.2. Expression Tester
- 1.3. Data
- 1.4. Applications
- 1.5. Configuration
- 1.6. Jobs
- 1.7. Generic Connector

1.1. Installation and Setup

A new desktop installer is now available to streamline installations for small teams and single-node setups. It removes the need to manually configure YAML parameters or use command-line arguments. The guided interface provides all required options with step-by-step instructions.

For more information, see *Ansys Minerva Installation Guide: Installing Ansys Minerva Using the Desktop Installer*.

1.2. Expression Tester

A new tool, **Expression Tester**, is now available, enabling you validate Mamba expressions before using them in features like verification plans, forms, and more. You can access the tool from the **User** → **Tools** menu available at the upper right.

It can also dynamically generate expressions to access elements from documents, such as forms, parameter documents, or reports.



For more information, see *Ansys Minerva User's Guide: Expression Tester*.

1.3. Data

- Previously, deleting a large set of files caused the Ansys Minerva user interface to become unresponsive. To resolve this, the deletion operation is now handled asynchronously using a conversion task with a conversion rule named **BackgroundTaskConverter**.

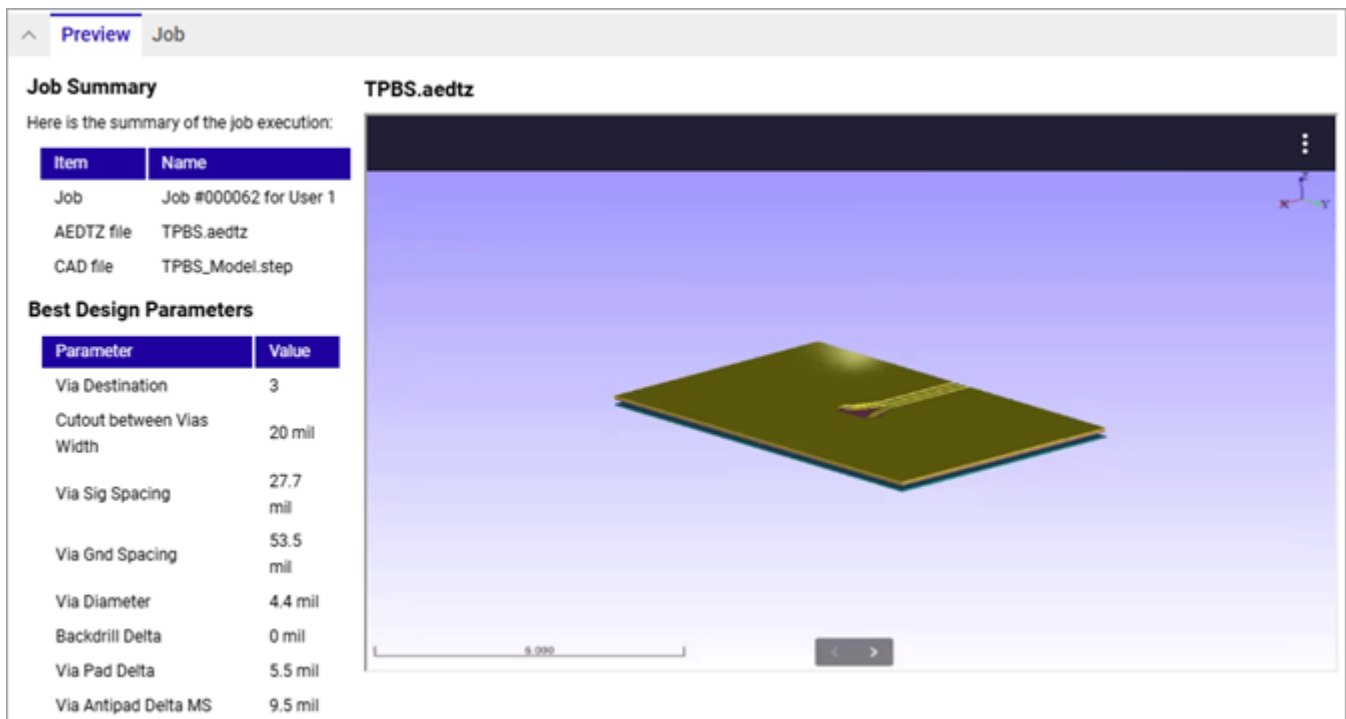
When you delete files, they are instantly removed from the interface, and a conversion task is created to process their deletion in the background.

Administrators can manage these tasks under **Contents** → **Administration** → **File Handling** → **Conversion Tasks** and perform regular cleanups as required.

- You can now view properties of parameter document elements in read-only mode.

1.4. Applications

The new **Output Form** option in the application definition lets you assign a form to summarize key information from the job output. When assigned, users can view the result summary in the **Preview** tab of the job.



The Ansys Mechanical APDL batch application included with the Ansys Minerva solution uses the **ans_JobPreview** output form. You can use this form as a reference when designing a new form for your application.

For more information, see *Ansys Minerva Configuration Guide: Defining Applications*.

1.5. Configuration

- The Forms 2.0 editor now includes the following new elements, allowing you to display advanced data on forms:

Category	Elements	Description
Tables	Table, Caption, Row, Cell, and Header Cell	Structure information in tabular format. See Table .
Information sets	Fact Set, Fact, Term, and Description	Add information sets like glossaries, metadata, or standards. See Fact Set (Information Set) .
3D images	WCAX Image, HOOPS Image, and AVZ Image	Add 3D model visualization files. See 3D Images (3D Model Visualization Files) .
Reusable element groups	Partial and Rener Partial	Reuse a group of elements across different contexts on a form. See Partial and Render Partial

- The default configuration for the **Vote** (ans_Vote) toolbar button has been removed from all ItemTypes except **Work Requests** and **Change Notices**. If you need this button for other ItemTypes, you now need to add it manually. For assistance, contact your technical support representative.

1.6.Jobs

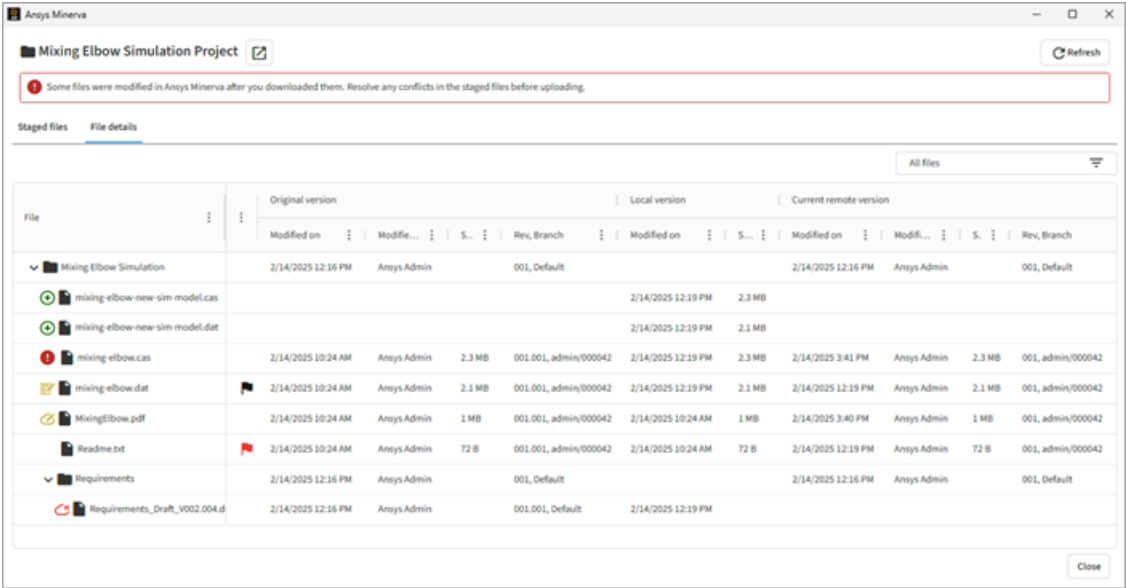
You can now submit batch jobs to Ansys HPC Platform Services (HPS). This enables you to seamlessly work with your on-premises HPC clusters as well as cloud-based resources. Ansys HPC Platform Services allows efficient use of compute resources and relieves you of having to choose a suitable and available resource for each job.

Ansys recommends using Ansys HPC Platform Services (HPS) for job submission. Ansys Distributed Compute Gateway DCG and Ansys Cloud Direct will be deprecated in future releases.

- For information about configuring an Ansys HPC Platform Services job server in Ansys Minerva, see *Ansys Minerva Configuration Guide: Defining Job Servers*.
- For information about deploying Ansys HPC Platform Services, see *Ansys HPC Platform Services Deployment Guide*.
- For information about Ansys HPC Manager, see *Ansys HPC Platform Services User's Guide*.

1.7.Generic Connector

- The new **File details** tab in the Generic Connector's staging window now provides detailed information about files in your local working directory, such as:
 - Changes to files, including changes made by other users on the server and any resulting conflicts.
 - Details of different versions of a downloaded file.
 - The claim status of your files on the server, with the option to claim files directly from the **File details** tab.
 - Unchanged files.



For more information, see *Ansys Minerva Generic Connector User's Guide*: [Viewing the Status of Your Files](#).

- The JSON output of the Generic Connector commands has been enhanced to include the following parameters:
 - `newly_downloaded`: After a download, it indicates whether an item was newly downloaded or already present in the working directory.
 - `conflict`: Indicates if an item has conflicts.

For more information, see *Ansys Minerva Generic Connector User's Guide*: [JSON Output Schema](#).

- The `--output` option is now optional for all Generic Connector commands. However, you need to specify it for some commands where output is essential to save the output information, such as when using **fetch-status** with the `--interactive` option set to `Console` or `None`.

2. Resolved Issues

For a list of issues that have been resolved in Ansys Minerva 2025 R2, see the [Ansys Minerva](#) section of the *Ansys Resolved Issues and Limitations*.

3. Known Issues and Limitations

All issues and limitations known at the time of release are listed in the [Known Issues and Limitations](#) section of the *Ansys Minerva Troubleshooting Guide*.

Issues discovered after release are listed in the [Ansys Minerva](#) section of the *Ansys Known Issues and Limitations*.

4. Documentation Enhancements

- A dedicated user's guide for Generic Connector is now available. See [Ansys Minerva Generic Connector User's Guide](#).
- The Forms 2.0 section in configuration guide has been improved to include a comprehensive walk-through video, task-oriented topics detailing the usage of individual Form 2.0 elements, and some form examples. See *Ansys Minerva Configuration Guide*: [Forms 2.0](#).
- A **Quick Start** section is now available for the Mamba Expression Language. It helps you quickly explore and understand commonly used expressions in Ansys Minerva, with examples and syntax explanations. See *Ansys Minerva Configuration Guide*: [Quick Start](#).

Distributed Compute Gateway (DCG)

The following topics provide an overview of what is new and improved in Distributed Compute Gateway 2025 R2:

1. [New Features and Enhancements](#)
2. [Resolved Issues](#)
3. [Known Issues and Limitations](#)

1. New Features and Enhancements

No enhancements were made to Distributed Compute Gateway in this release.

2. Resolved Issues

For a list of issues that have been resolved in Distributed Compute Gateway 2025 R2, see the [Distributed Compute Gateway \(DCG\)](#) section of the *Ansys Resolved Issues and Limitations*.

3. Known Issues and Limitations

All issues and limitations known at the time of release are listed in the [Troubleshooting](#) section of the *Distributed Compute Gateway User's Guide*.

Issues discovered after release are listed in the [Distributed Compute Gateway \(DCG\)](#) section of the *Ansys Known Issues and Limitations*.

optiSLang

The following sections contain release information for optiSLang 2025 R2:

- 1. General
- 2. Licensing
- 3. User Interface
- 4. Connectors
- 5. Algorithms

1. General

PyoptiSLang 1.0

PyoptiSLang 1.0 is now embedded into the automated installer:

- Robust design with optimization (RDO) workflow w/ pyoSL
 - New optiSLang Collab landing page linking to the [pyoptiSLang "RDO workflow example"](#)
 - Driven and adopted by Ansys products Disco, Fluent, Raptor, Speos, Solutions



About this project

Repositories where the teams can collaborate with other BUs to work on optiSlang integrations

[Wiki / Introduction](#)

All you need to start

optiSlang workflow generation and automation

[optiSlang](#)

[SDO workflow inside your tool](#)



Python based integration plugins for optiSlang

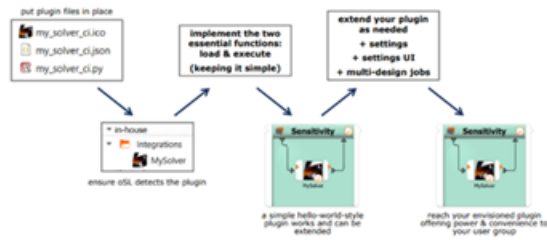
[Getting started with python based optiSlang integrations](#)

[Python based optiSlang integrations - Ansys optiSlang Documentation](#)

[Python based optiSlang integrations - Demonstrator Description](#)

[Demonstrations](#)

How to provide a plugin?



SSH Remote Profile

SSH Remote Profile now supports using a shared drive:

- While running remotely, data is sourced from the optiSlang project located on the shared drive
- No file upload and download is necessary

Supported Versions

Updated [support for integrated versions](#), for example, Abaqus ETK up to version 2025.

Documentation

The following documentation enhancements were made:

- Added Python documentation to the Ansys developer portal
- Added a "What's New" document to highlight updates to optiSlang features.
- Consolidated some documents into the [User Guide](#):

- Moved the [optiSLang Web Service User Guide](#) document into the User Guide under the new "Legacy" section.
- Moved the [Excel Plug-In](#) document into the User Guide.
- Moved the [Beta](#) document into the User Guide.
- Moved the [oSP3D Processing](#) document into the User Guide.
- Moved the *optiSLang Web Service Install Guide* into the optiSLang Installation Guide under the new "Legacy" section.
- Renamed the [Interfaces and Customization](#) guide to "optiSLang Customization, API, and Remote Control".

2. Licensing

DSDXM license

Basic functionality support with [DSDXM license](#) has been added:

Note:

DesignXplorer is deprecated. The DSDXM license enables comparable functionality for Correlation analysis, Design of Experiments, Response Surface modeling, and Optimization.

- Restrictions with the DSDXM license:
 - Wizards: Restrictions in Sensitivity, Optimization, and Robustness wizards
 - MOP/RSM: can only run RSM, no competition, no variable reduction, reduced list of meta-models
 - DOE: restrictions on Sensitivity wizard, dynamic sampling, and start design generation
 - Optimization:
 - limited to Mixed-Integer Sequential Quadratic Programming (MISQP) and Evolutionary Algorithm (EA)
 - other wizard and settings restrictions
 - EA: no advanced settings

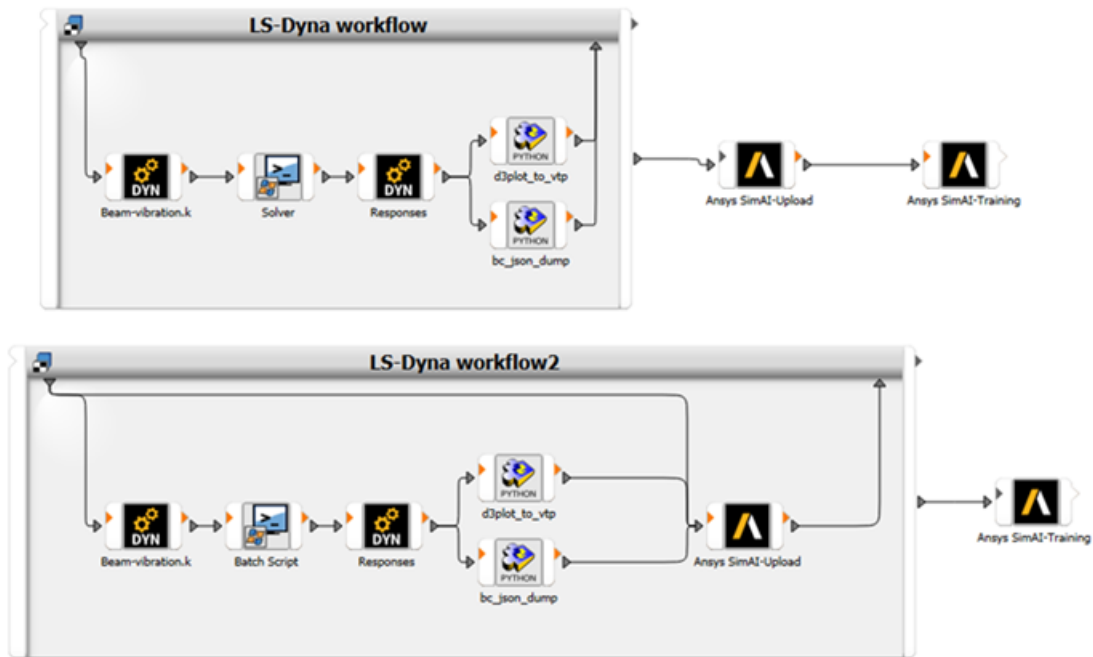
3. User Interface

The optiSLang user interface and related features have been updated:

Ansys SimAI Integration

Ansys SimAI Integration has been added:

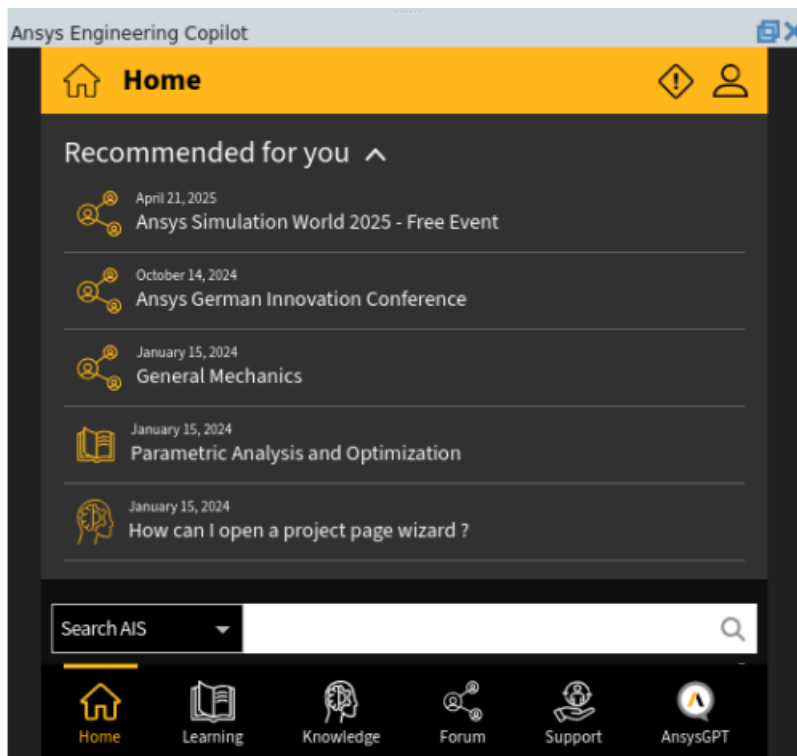
- Two nodes are available: [SimAI Upload \(Beta\)](#) and [SimAI Training \(Beta\)](#)
- You can generate training data and then upload one design at a time or all the designs at once.
- You can then run Ansys SimAI on the training data.



Ansys Engineering Copilot

[Ansys Engineering Copilot integration](#) (Beta) has been added inside optiSLang:

- Available in Ansys optiSLang Postprocessing and Ansys optiSLang GUI
- Opens as a dock widget
- Lets you search Ansys Innovation Space, knowledge articles, forum posts, and AnsysGPT™.



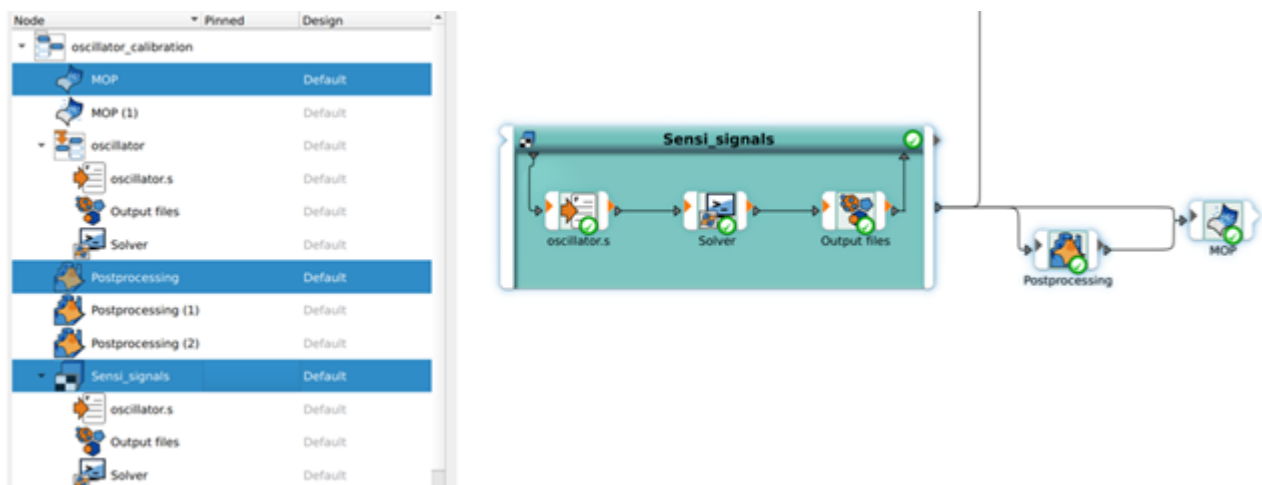
Ansys Minerva

The Ansys Minerva connector has been updated to use the Ansys Minerva CLI interface.

Project Tree/Scenery

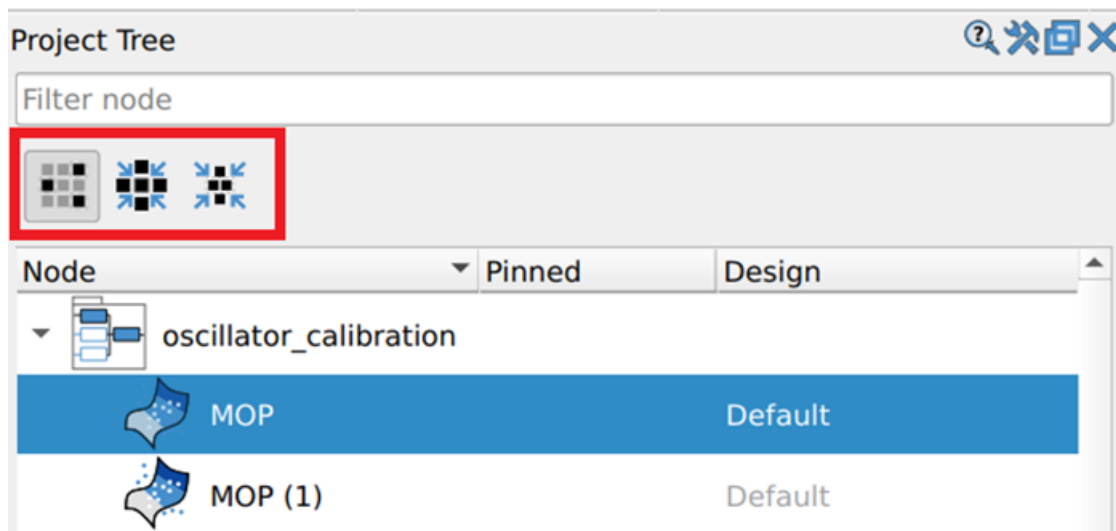
The [Project Tree/Scenery](#) have been updated:

- You can now select a node or system in the Project Tree and the selection will be reflected in the Scenery, and vice versa.



- The context menu in the Project Tree and in the Scenery is now the same in both locations.

- New options to focus the scenery have been added:

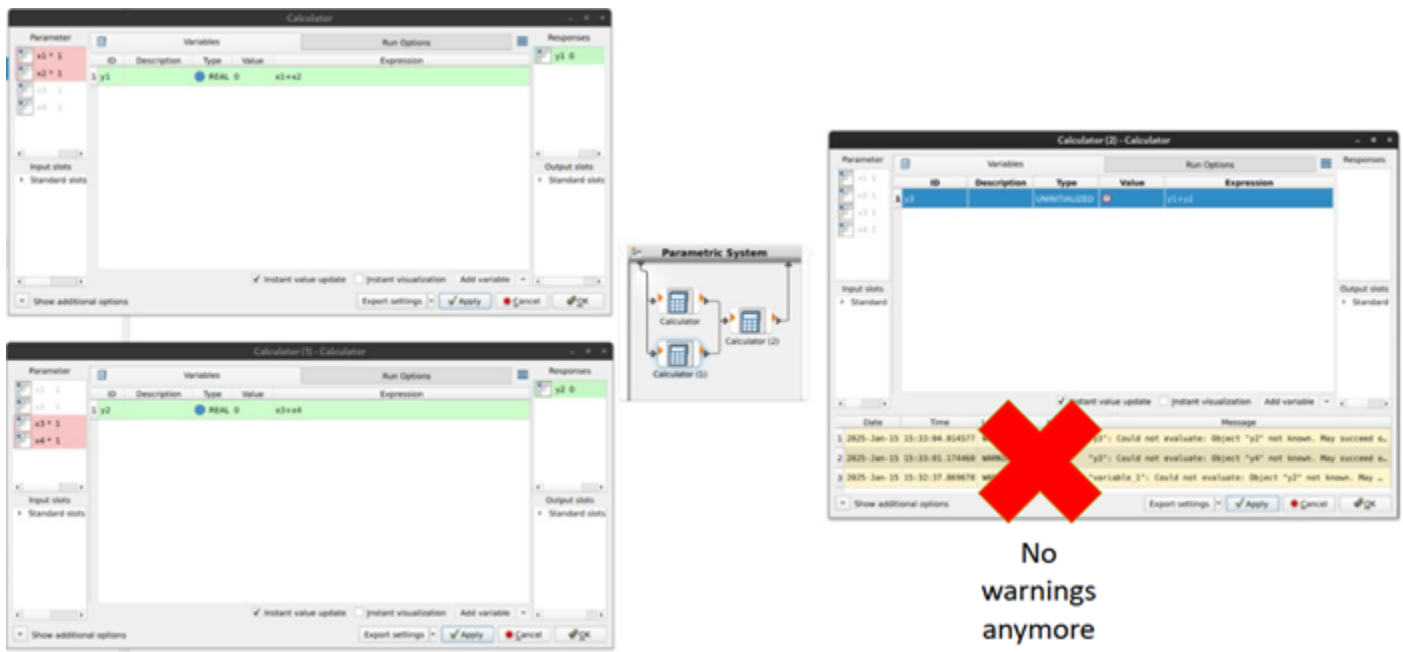


-  Highlight selected nodes - On selection of nodes, highlight selected nodes and "fade out" non-selected nodes in workflow
-  Focus on current node - On change of current node in project tree, centers the scenery on that node
-  Fit view on selected nodes - Zoom and pan the view to fit the selected nodes in project tree

Auto-propagation of Responses

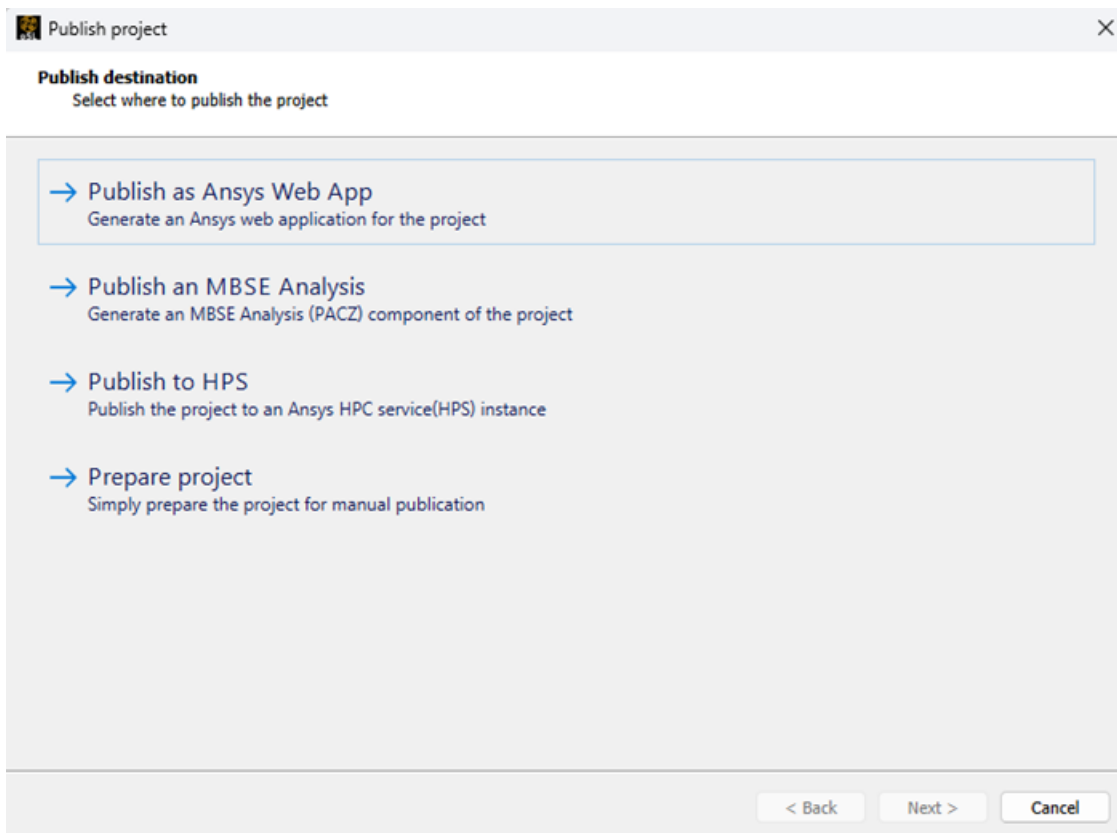
Response values in the **Calculator** node and **Variables** tab in connector nodes now auto-propagates from parent nodes during setup.

- Expressions that use these can now be defined/evaluated during setup time.
- No more warnings during setup time. Responses registered in parent parametric systems are already known to the calculator.
- Auto-complete and preview (**Ctrl+Space**) provides the responses.



Publish Wizard

A new [Publish project wizard](#) has been added. This wizard gives a unified UI for the following capabilities:



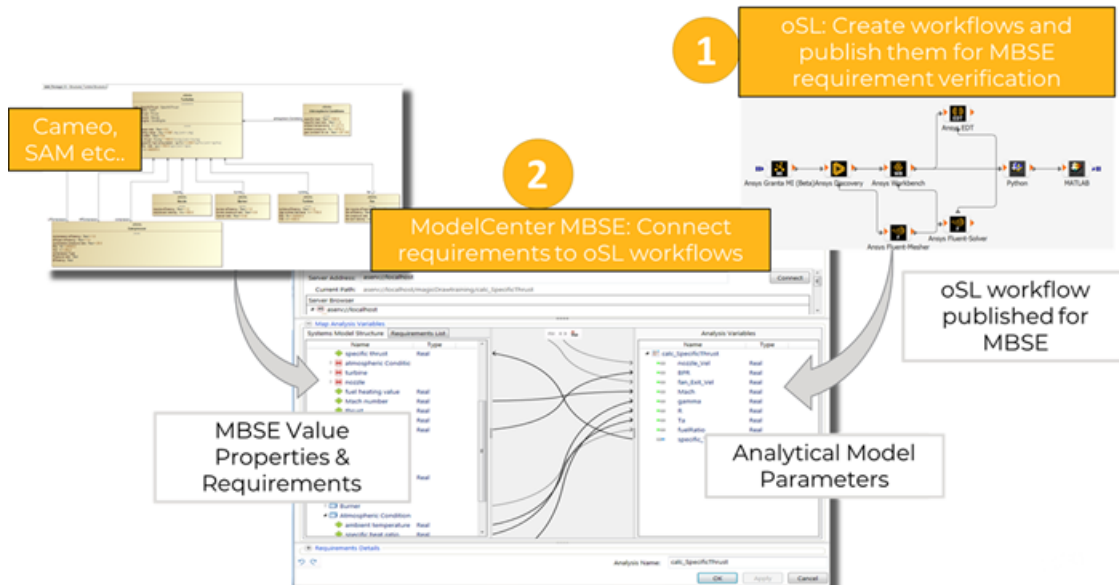
- [Publish as Ansys Web App](#)

- Ansys Web Apps can be also be published directly to [Ansys Minerva](#).
- NEW - [Publish as MBSE component \(PACZ\)](#) (p. 228)
- NEW - [Publish to HPS \(Beta\)](#) (p. 228)
- Prepare project

Publish as MBSE component (PACZ)

optiSLang workflows can now be [published as an MBSE component \(PACZ\)](#):

- Create workflows in optiSLang and publish them for MBSE requirement validation
- Use ModelCenter MBSE to connect optiSLang workflows to Systems Architecture Modeling (SAM) tools (such as Ansys SAM and Cameo), which provide MBSE value properties and requirements



Publish to HPS (Beta)

[Publish to HPC Platform Services \(HPS\)](#) (Beta) has been added:

- Easy use of HPC resources
- Reduction of scripting effort

Execution settings

[Execution settings](#) have been unified:

- Setting environment variables is now available in the following nodes:
 - Merged and Replaced in: Workbench, optiSLang, LS-Extractor, Add
 - Merged in: Discovery, Speos, HPS, Python node

- Python based Add-ins (tab activation only)
- Updated the [Run Options](#) tab:
 - Merged the execution options:
 - Removed tab "Execution settings"
 - Introduced "Distinct working directory"
 - Created a new arrangement of options in groups:
 - General
 - Execution
 - Solver specific
 - Remote execution
- Other improvements:
 - Show slot values as tooltips in slot boxes
 - Definition of custom input values now in the Input slots box
 - Removed "Additional options" if Input slot box exist
 - Deprecated Input/Output files tabs for Process/Script nodes
- Adjusted option names:
 - Delay before start → Delay before execution
 - Maximum runtime → Maximum execution runtime
 - Maximum in parallel → Maximum simultaneous executions
 - Use remote profile → Remote profile
 - Polling time interval → Polling interval
 - Retry connection timeout → Reconnect timeout
- GUI ready nodes:
 - Process node
 - Script nodes (Batch, Bash, Perl, Python)
 - Python node

4. Connectors

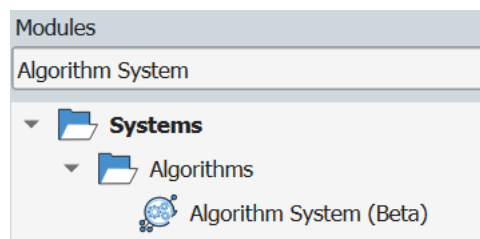
- [Thermal Desktop node](#) has been moved out of Beta.
- [HPS node](#) has been moved out of Beta.
- [Signal Competition node](#) has been moved out of Beta.
- [NX node](#) now supports exporting to PARASOLID, STEP, and IGES.
- [Ansys LS-DYNA Output node](#) now supports multihistories and multiresponses.
- [Data mining node](#) now supports functions to get all feasible or infeasible designs.
- [ETKnD -- RST to VTU/VTP conversion node \(Beta\)](#) has been added.
- [ConceptEV node \(Beta\)](#) has been added - a connector for the new Ansys tool
- [MOP3 Node \(beta\)](#) has been updated with [Custom Meta-Models](#), [Custom Quality Measures](#), and [Signal Model Competition](#).
- [Remote Submission](#): Ansys Distributed Compute Services (DCS) has been deprecated and is no longer supported as of July 2025. SSH and Ansys HPC Platform Service (HPS) are the supported remote services.

5. Algorithms

Algorithm System (Beta)

Algorithm system (beta) has been adopted:

- Design results are now available in the Algorithm System
- Algorithm instance ID are synced between UI and backend with "live update" during run
- You configure which state to track in the project tree



Id	Status	Evaluation status	Parameters		Responses
			x	y	z
> 1	Evaluated	Not evaluated	-0.9900000000000001	-0.9900000000000001	-1.9800000000000002
> 2	Evaluated	Not evaluated	0.25	-0.53	-0.28
> 3	Evaluated	Not evaluated	0.0900000000000008	-0.27	-0.1799999999999994
> 4	Evaluated	Not evaluated	0.55	-0.0899999999999997	0.4600000000000001
> 5	Evaluated	Not evaluated	-0.1099999999999997	0.8700000000000001	0.7600000000000001
> 6	Evaluated	Not evaluated	-0.35	-0.25	-0.6
> 7	Evaluated	Not evaluated	0.1499999999999999	0.8300000000000001	0.98
> 8	Evaluated	Not evaluated	0.8300000000000001	0.6499999999999998	1.48
> 9	Evaluated	Not evaluated	0.4299999999999994	-0.71	-0.28
> 10	Evaluated	Not evaluated	0.75	0.9700000000000001	1.7200000000000002

One-Click Optimization (OCO)

In [One-Click Optimization \(OCO\)](#), the number of initial design evaluation options was updated to allocate fewer designs to the initial sampling and more designs to the optimizers.

Calculator functions

New Calculator functions were added:

- [Standard normal distribution - CDF - phi\(\)](#)
- [Standard normal distribution - Inverse CDF - phi_inv\(\)](#)
- [Determinant - det\(x\)](#)
- Identity matrices:
 - [identity\(n\)](#)
 - [identity\(n,m\)](#)

Other

- "Bring your own AI" has been moved out of Beta
- "FMU export of DFFN (beta)" is now "FMU export of DFFN in MOP3 (Beta)"

Ansys Platform Products

Release notes are available for the following Ansys Platform products:

[ACT \(p. 235\)](#)

[Ansys Dynamic Reporting \(p. 237\)](#)

[CAD Integration \(p. 241\)](#)

[DesignXplorer \(p. 247\)](#)

[RSM \(p. 249\)](#)

[System Coupling \(p. 251\)](#)

[Workbench \(p. 255\)](#)

ACT

No enhancements were made to ACT in 2025 R2.

Ansys Dynamic Reporting

This document covers the Ansys Dynamic Reporting release notes for versions 2025 R1 and later. For previous release notes, see the Ansys Dynamic Reporting section in the relevant version of the [EnSight release notes](#).

The following list highlights the new features and enhancements for Ansys Dynamic Reporting in 2025 R2.

- Added support for [3D plots and polar plots](#).
- Added support for [OBJ and SCDOX](#) 3D formats.
- You can now [export and load templates as JSON files](#).
- Added [default report templates](#).
- Expanded the [custom web components](#) feature to add more report templates and allow for more customization.
- You can now display [aggregate filters](#) in templates.
- You can now display a [dynamic list of items](#) for each template leaf.
- The workflow for [conditional formatting settings](#) has been improved.
- You can now display the [table of contents](#) on the side of the report.
- You can now set the [slider type](#) to either a standard slider or a drop-down menu.
- Add option to dynamically change the data source and some rendering parameters for the 3D viewer.
- Fully released the following report templates:
 - [Data Filter](#)
 - [PowerPoint](#)
- Added a new query type to execute numerical comparisons in tags (API only).
- Made performance improvements in API for creating and modifying report templates.
- Improved API length checks for strings.

Ansys Viewer

The following sections contain release information for Ansys Viewer 2025 R2:

1. [Advisory: Ansys Viewer End of Life](#)
2. [New Features and Enhancements](#)
3. [Resolved Issues and Limitations](#)
4. [Known Issues and Limitations](#)

1. Advisory: Ansys Viewer End of Life

Ansys Viewer 2025 R1 will be the final version of Ansys Viewer released by Ansys. As of this release, no further updates or bug fixes will be provided.

As of the next Ansys release (2025 R2), Ansys Viewer will be removed from the Ansys Installation program, and support for Ansys Viewer will end.

If you use Ansys Viewer for graphics visualizations, Ansys encourages you to transition to alternative tools or workflows. If you have any questions or need assistance, please contact Ansys support.

Note that this advisory is *not* referring to the *Ansys Help* viewer, which provides access to Ansys help documentation for customers without internet access.

2. New Features and Enhancements

No enhancements were made to Ansys Viewer in this release.

3. Resolved Issues and Limitations

See the [Resolved Issues and Limitations](#) document for information on previous limitations and key defects resolved in the current release.

4. Known Issues and Limitations

All issues and limitations known at the time of release are listed in the [Known Issues and Limitations](#) section of the *Ansys Viewer User's Guide*.

CAD Release Notes

This section summarizes the new features in CAD Integration Release 2025 R2.

For more information, see the [CAD Integration](#) section of the Ansys Help.

Rocky Linux 8.10 and 9.4 Support

The following interfaces are supported by Ansys:

- ACIS (Reader)
- AutoCAD (Reader)
- CATIA V4 (Reader)
- CATIA V5 (Reader)
- CATIA V6 (Reader)
- Creo Parametric (Reader)
- IGES (Reader)
- Inventor (Reader)
- JT (Reader)
- Monte Carlo N-Particle
- NX (Reader and Plug-In)
- Parasolid (Reader)
- Rhinoceros (Reader)
- Solid Edge (Reader)
- SOLIDWORKS (Reader)
- STEP (Reader)

Ubuntu LTS 20.04 and 22.04 Support

The following interfaces are supported by Ansys:

- ACIS (Reader)
- AutoCAD (Reader)

- CATIA V4 (Reader)
- CATIA V5 (Reader)
- CATIA V6 (Reader)
- Creo Parametric (Reader)
- IGES (Reader)
- Inventor (Reader)
- JT (Reader)
- Monte Carlo N-Particle
- NX (Reader)
- Parasolid (Reader)
- Rhinoceros (Reader)
- Solid Edge (Reader)
- SOLIDWORKS (Reader)
- STEP (Reader)

Geometry Interface Update for New CAD Releases

Geometry interfaces are updated to support new CAD releases including:

- AutoCAD 2024 (Reader, Linux and Windows)
- AutoCAD 2026 (Plug-In, Windows only)
- CATIA V4 4.2.5 (Reader, Linux and Windows)
- CATIA V6 2024x (Reader, Linux and Windows)
- Creo Parametric 11.0 (Reader, Linux and Windows)
- Inventor 2025 (Reader, Linux and Windows)
- Inventor 2026 (Plug-in, Windows only)
- JT 10.9 (Reader, Linux and Windows)
- NX 2412 (Reader, Linux and Windows)
- Rhinoceros 8 (Reader, Linux and Windows)
- Solid Edge 2025 (Reader, Linux and Windows)
- Solid Edge 2025 (Plug-In, Windows only)

-
- SOLIDWORKS 2025 (Reader, Linux and Windows)

For detailed version support information, see CAD Integration> Geometry Interface Support in the [CAD Integration](#) section of the Ansys Help.

Information about past, present and future CAD, operating system and platform support is viewable via the Ansys, Inc. website (Customer Care > Support> More Support > Platform Support).

Distributed Compute Services (DCS)

No enhancements were made to DCS in 2025 R2.

DesignXplorer

DesignXplorer is deprecated. Ansys recommends using optiSLang containers.

The DesignXplorer license (dsdxm license feature) enables comparable functionality for Correlation Analysis, Design of Experiments, Response Surface Modeling, and Optimization. You can reuse your design points generated by DesignXplorer in optiSLang by using the **Export DX Data** and **Import DX Data** menu options in the Ansys Workbench optiSLang Extension. For more information, see [optiSLang Extension User Interface Reference](#).

Remote Solve Manager (RSM)

The following sections contain release information for Ansys Remote Solve Manager 2025 R2:

- 1. [New Features and Enhancements](#)
- 2. [Issues Resolved in this Release](#)
- 3. [Known Issues and Limitations](#)

1. New Features and Enhancements

No enhancements were made to Remote Solve Manager in this release.

2. Issues Resolved in this Release

For a list of issues that have been resolved in Remote Solve Manager 2025 R2, see the [Remote Solve Manager \(RSM\)](#) section of the *Ansys Resolved Issues and Limitations*.

3. Known Issues and Limitations

All issues and limitations known at the time of release are listed in the [Known Issues and Limitations](#) section of the *Remote Solve Manager User's Guide*.

Issues discovered after release are listed in the [Remote Solve Manager \(RSM\)](#) section of the *Ansys Known Issues and Limitations*.

System Coupling

The following features have been added to the Ansys System Coupling™ 2025 R2 release:

1. Graphical User Interface
2. Participants
3. Mapping Accuracy
4. Solver Precision
5. Expressions
6. User-Friendly Mapping
7. Terminating Participant Processes
8. Reference Frames and Instancing
9. System Coupling Infrastructure
10. Data Model and Commands
11. Participant Library
12. Known Issues and Limitations

1. Graphical User Interface

The following improvements have been made to System Coupling's graphical user interface (GUI):

- More options have been added to the [toolbar buttons](#), [Menu Bar](#), and [context menu](#).

2. Participants

The following improvements have been made to System Coupling's participant coupling:

- Added Ansys Rocky as a solver in the System Coupling ecosystem.
 - Coupling occurs on walls. Rocky can receive displacements and temperatures as inputs. Rocky outputs forces and heat flows.
 - Two-way Rocky and Mechanical structural coupling.
 - Two-way Rocky and Mechanical thermal coupling.
- Updated the Maxwell magnetostatic solver with additional variables available for coupling. Inputs are displacements and rotations (single-valued parameters). Outputs are forces and torques (also single-valued parameters).
- Added ability to model two-way impact of electromagnetic forces on the electrical switchgear mechanism for electric arc modeling.

- Can now couple Fluent or AEDT and rigid body solvers (for example, Ansys Motion FMU) to perform two-way fluid-structure interaction.
- FMU proxy participants now support real, integer and string FMU parameters (attributes).

3. Mapping Accuracy

You can expect different numerical results due to the following improvements to System Coupling's mapping capabilities:

- Accuracy improvements when mapping node-based extensive data between surfaces or from point cloud to surface.
- Accuracy improvements to volume element-to-element profile-preserving mapping.
- Accuracy improvements to surface element to point cloud mapping, especially on the non-overlapping points.
- Improved the non-overlap handling behavior for surface to point cloud mapping. The previous behavior was that point cloud points far from the surface were being assigned zero values despite being reported as mapped. The new behavior depends on if the source is extensive:
 - If source data is extensive, non-overlapping target values continue to receive zero values (same behavior) but are indicated as unmapped.
 - If the source data is intensive, non-overlapping target point cloud points are mapped from the nearest surface points, and continue to be considered as mapped.
- Accuracy improvements when mapping profile-preserving data on surfaces from nodes to elements.

4. Solver Precision

The CFX participant now defaults to a double precision solver. This is a change in behavior from previous releases (defaulting to single precision).

Also added global solver precision control. This forces Fluent and CFX to run with the same precision (single or double).

5. Expressions

The following improvements have been made to System Coupling's expressions:

- Added [reduction expressions](#).
- Added ability to calculate Sum, Minimum, Maximum, and Average of field variables/expressions.
- Added [vector dot and cross](#) operations.

6. User-Friendly Mapping

- Added a new [user-friendly mapping workflow](#).
 - Provides interactive setup with mesh and mapping visualization and the ability to adjust some setup and re-map interactively.
 - Can be used to accurately map field data, for example between meshes stored in files of different formats.
 - Scripting is not required.
 - Leverages embedded viewer (EnSight) for visual feedback about mapping.
- Added mapping of mode shapes from Mechanical modal analysis to CFD blade flutter analysis for aerodamping. Interactive setup with mesh and mapping visualization. Ability to adjust some setup and re-map interactively

7. Terminating Participant Processes

Created a [script](#) to automatically kill all processes spawned by System Coupling.

Also created a GUI [menu option](#) to surgically kill only participant solver processes. It is a more drastic measure than aborting or interrupting the solve. It immediately stops the simulation.

These options help you clean up orphaned processes if something goes wrong in one of the solvers and simulation crashes.

8. Reference Frames and Instancing

Reference frames and instancing are now applied in a different order when both the Instancing and ReferenceFrame objects are applied to the same side of an interface. The previous behavior applied the reference frame first, which made it difficult to determine the rotation axis for the instancing object since you would need to consider instancing of the transformed geometry. Now instancing is applied first, and the instancing axis is relative to the original geometry configuration, which makes setting up the instancing and reference frames more intuitive.

9. System Coupling Infrastructure

The following improvements have been made to System Coupling's infrastructure:

- On Windows, System Coupling now uses [local host](#) for solver connections by default.
- Ability to add [environment variables](#) for individual participants.
- Handling of data transfers between implicit and explicit participants has been corrected. Data from explicit participants are now available in an implicit participant prior to entering the iteration loop.

10. Data Model and Commands

A list of the changes made to System Coupling's data model and commands is available in the [Migrating Command-Line System Coupling Scripts](#) section of the *System Coupling User's Guide*.

11. Participant Library

Extensions to System Coupling's Participant Library APIs improve usability and broaden the range of functionality available to coupling participants. These changes are listed in the **Changelog** section of the [System Coupling Participant Library](#) documentation.

12. Known Issues and Limitations

All issues and limitations known at the time of release are listed in the [Known Issues and Limitations](#) section of the *System Coupling User's Guide*.

Workbench

The Ansys Workbench platform offers many new features and enhancements. Areas where you will find changes and new capabilities include the following:

1. General Workbench Enhancements
2. Engineering Data Workspace
3. Injection Molding Data

1. General Workbench Enhancements

The following general enhancements were made for 2025 R2:

- The [User String](#) field has been released from beta.
- **Substructure Analysis.** From the **Workbench** Project Schematic, you can now link the **Solution** cell of a [Substructure Generation](#) analysis to the **Model** cell of a supported downstream system to automatically create an [Imported Condensed Part](#) object, completely defined, in the downstream system. This process automatically generates two [Exported Condensed Part](#) (*.cpa) files and provides you with the option to select which file, either the Use Pass or both the Use Pass and the Expansion Pass, to create the Imported Condensed Part.
- New automation for Sherlock workflows:
 - New automatic renaming option for model assembly workflows. In the context of Sherlock workflows, disables Mechanical renaming for the assembly.
 - Automatically keep modal results for Sherlock random vibration analysis in Mechanical.

2. Engineering Data Workspace

For Release 2025 R2, the following enhancements have been made to the Engineering Data workspace:

- **Crack Growth - Paris' Law.** You can now directly choose the units of measurement for **Length per Cycle Units** and **Stress Intensity Factor Units** instead of using reference units. This allows you to enter experimental material data directly in your preferred unit system. Workbench and the solver automatically convert these values to the current unit system.
- The following material models for fatigue crack growth analyses are now available:
 - **Tabular Fatigue Law**
 - **Forman Equation**
 - **Walker Equation**

- **NASGRO Equation V3**
- **NASGRO Equation V4**

3. Injection Molding Data

For Release 2025 R2, there are no new feature updates for [Injection Molding Data](#).

Embedded Software Products

Release Notes

Release notes are available for the following Ansys embedded software products:

[Scade One \(p. 259\)](#)

[SCADE Suite \(p. 263\)](#)

[SCADE Display \(p. 267\)](#)

[SCADE Test \(p. 271\)](#)

[SCADE Architect \(p. 275\)](#)

[SCADE LifeCycle \(p. 279\)](#)

[SCADE Solutions for ARINC 661 Compliant Systems \(p. 283\)](#)

[SCADE Avionics Package \(p. 287\)](#)

Scade One Release Notes

The following sections present the new features and enhancements for Scade One 2025 R2 release.

[1. New Features and Enhancements](#)

[2. Limitations/Known Issues](#)

1. New Features and Enhancements

This section describes the new features and enhancements introduced in SCADE Suite 2025 R2 release.

IDE

- Automated workflow of debugging operators:
 - Start a debug session from an operator
 - Keep or discard changes to data source files at the end of a debug session
- Services for editing declarations in module interfaces and bodies
- Quick navigation: between a module interface and module body, between a declaration and a definition

Language and code generation

- Improved performance of generated code
- Fixed code generator limitations:
 - Mix of weak and strong transition types
 - **unless** condition of **forward**

PyScadeOne

- Jobs reading and execution
- New Python wrapper service: wrap generated code as Python code
- Python libraries for model creation and update

Tools

- Integration with the MBSE portal and SAM Enterprise:

- Protocol handlers to call the Scade One IDE from the portal
- Ability to push/pull projects to/from the portal
- Prototyping mode, included in SAM Enterprise
- Ansys Engineering Copilot: AI-powered virtual assistant
- Job Launcher

2. Limitations/Known Issues

This section describes the limitations and workaround solutions applying to Scade One 2025 R2 release.

Language/Code Generator

The following constructs are not supported:

- **forward** with a return clause specifying both **last** and **default** expressions
- **forward** with an array return clause specifying a **last** expression
- Abstract types, that is, types with a declaration (in the module interface) and a definition (in the module body)

Modeler

The following language features/construct are not supported by the modeler:

- Comments entered at the end of a text (for example, at the end of a global declaration) are lost after save/load.

Workaround: Put the comments before the declaration.

- Variant, char types, and groups are not supported in the signal editor.

Debugger

- Variant, char types, and groups are not supported during interactive debug.
- Not possible to select the instance for a **forward** in a textual operator, for an anonymous operator under an iterator in both textual and graphical operators.

SCADE Importer

- Synchro transitions are not correctly imported if there is a state that is both initial and final. A warning is raised in that case.
- There may be overlaps in the generated diagrams: using the `-diagram_ratio` option can reduce these overlaps.

- The output model may be incorrect in the following cases (because of changes in the language):
 - The input model contains anonymous polymorphic structures (ex : {a: 'T'}).
 - The input model contains imported numeric types.
 - The input model contains public declarations whose definition depends on private imported types, private enumerated types, or values of a private enumerated type.
 - The input model contains unsupported combinations of **activate** and **restart**.
 - The input model contains **l xor**, **cast** or n-ary operators in size expressions.
 - The model contains signals declared in a state and emitted in a weak transition of that state.
 - Partial iterators applied on **restart** or **activate .. every** operators are imported into a correct Scade One model, but the behavior is not preserved. A warning is raised in that case.

Workaround: Fix the model manually after import, using the diagnostics raised by the modeler.

PyScadeOne

Consult the [PyScadeOne](#) documentation to get information about limitations for each service.

SCADE Suite Release Notes

The following sections present the new features and enhancements for SCADE Suite 2025 R2 release.

- 1. [New Features and Enhancements](#)
- 2. [Certification Credits](#)
- 3. [Limitations and Workarounds](#)
- 4. [Migration Information](#)
- 5. [Tools Version Compatibility](#)

1. New Features and Enhancements

This section describes the new features and enhancements introduced in SCADE Suite 2025 R2 release.

Renewed certification credits for:

- SCADE Test verification tools (see [Certification Credits \(p. 271\)](#))
- SCADE LifeCycle Reporter (see [Certification Credits \(p. 279\)](#))

SCADE Suite UX Improvements

- Start Simulator from SCADE Test results
- Copy full path of a file from title tab
- Copy/move field of structures to other structures
- Goto type definition added for Array elements
- Ctrl+A shortcut added in some missing areas

SCADE Test Integration

For details, consult [SCADE Test Release Notes \(p. 271\)](#).

SCADE LifeCycle Integration

For details, consult [SCADE LifeCycle Release Notes \(p. 279\)](#).

2. Certification Credits

The SCADE Suite 2025 R2 release includes the following tool with certification credits:

- **SCADE Suite KCG 6.6.4** and **SCADE Suite KCG 6.6.2** for qualification under DO-178C/DO-330 up to TQL-1 and for certification under ISO 26262 up to ASIL D, IEC 61508 up to SIL 3, or EN 50128 up to SIL 3/4
- **SCADE ACG 2.6** for certification under ISO 26262 up to ASIL D
- See also [Certification Credits \(p. 271\)](#) for SCADE Test and [Certification Credits \(p. 279\)](#) for SCADE LifeCycle

3. Limitations and Workarounds

This section describes the limitations and workaround solutions applying to the SCADE Suite 2025 R2 release.

Limitation in Simulink Importer

- Import does not support tunable popup parameters set using a referencing enumeration class.

Limitation in Code Generation for FMUs

- When using FMU states, any modification of model requires to save FMU states.

Limitation in Python Scripts

- Some Python extension modules (Python modules implemented in C) like numpy or lxml are not working in SCADE environment.

Limitation in SCADE Suite Modeling

- Floating windows do not operate properly when a SCADE Architect project is loaded at the same time as SCADE Suite: close and reopen SCADE Suite to resolve the issue.

Limitation in SCADE Suite Simulation

- Ada language pragmas referenced in SCADE Suite KCG “pragma” pragma may potentially not be supported for Ada code simulation.

Limitations in Code Integration

- [Ada] Provided Targets and Adaptors do not support Ada. Ada dedicated utilities can be developed using Python API.
- [C] Imported types at root level are not supported when generating wrapping code for Ansys Twin Builder.

Limitation in Multicore Code Generator

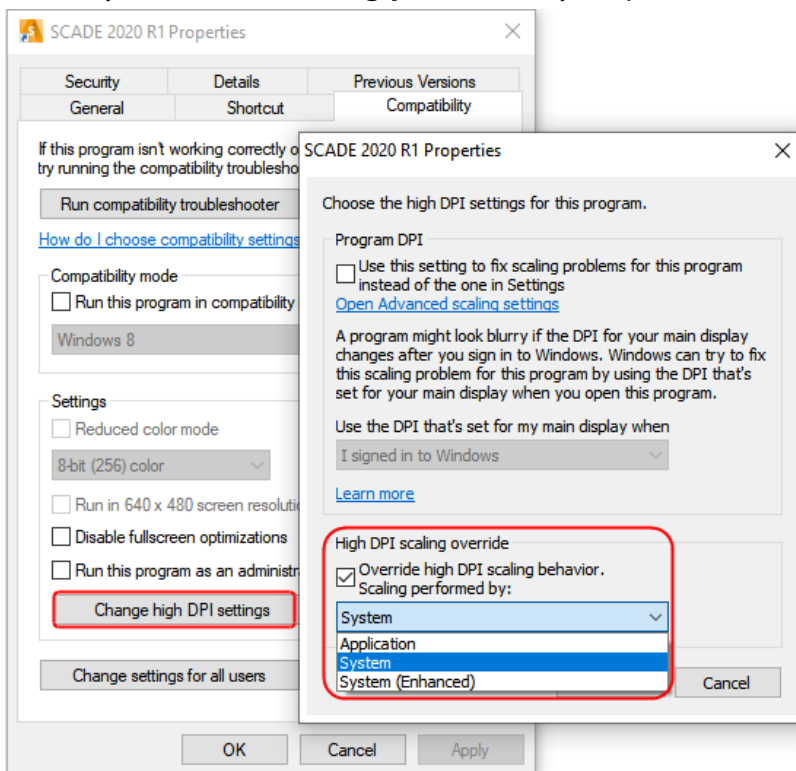
- With clocked tasks, generated code may contain cyclic dependency between methods, which prevents from integrating it.

Limitation in SCADE Applications

- SCADE applications may not scale well (with symptoms like tiny icons and UI elements) on high-DPI (high-dots-per-inch) displays, depending on the display scaling factor selected for the user's Windows display settings.

Workaround*: users can let Windows automatically manage the scaling of SCADE applications. This may result in applications being blurry, but scaled correctly.

1. Right-click the SCADE product binary (scade.exe) in File Explorer (or Desktop shortcut) to open Properties. On Compatibility tab, click **Change high DPI settings**, under High DPI scaling override section, check the **Override high DPI scaling behavior** option.
2. Select **System** for the **Scaling performed by** drop-down.



* This is only applicable to Windows 10 version "1703" (also called "Creators Update", delivered in April 2017) and subsequent ones, and can be applied to all SCADE versions. The same operation can be applied to the vcs.exe binary in the same folder.

Limitation in PDF User Documentation

- Hyperlinks in PDF documents opened in some web browsers may not work, preventing navigation between documents.

Workaround: set Adobe Acrobat as default PDF reader.

4. Migration Information

SCADE Suite 2025 R1 to 2025 R2

- No format change: no project migration issue

SCADE Suite 2024 R2 to 2025 R1

- No format change: no project migration issue

SCADE Suite 2024 R1 to 2024 R2

- No format change: no project migration issue
- Support for migrating Scade 6.4 projects to Scade 6.6 projects is no longer available. Loading Scade 6.4 models is no longer supported. Please contact your Ansys representative or Support if you have such need.
- In SCADE Suite Metrics and Rules Checker, the rule SCADE_R001 (for detecting non-coverable points) is replaced by a set of design and methodological rules. In case, you were using such a rule in your projects, it is recommended to launch Model Coverability analysis in SCADE Suite. If need be, the rule can be requested from support.

SCADE Automotive Package 2024 R2 to 2025 R1

- Migration to extended AUTOSAR configuration done automatically upon project loading - No specific issue: see [SCADE Automotive Package \(each release\) \(p. 266\)](#)

SCADE Automotive Package 2024 R1 to 2024 R2

- No format change: no project migration issue: see [SCADE Automotive Package \(each release\) \(p. 266\)](#)

SCADE Automotive Package (each release)

- SCADE Architect projects referencing libraries and/or configurations must be properly migrated. The migration of projects with libraries can be automated by a script distributed within SCADE installation and located at \SCADE\scripts\Migration\Architect.

For migration information related to previous versions, contact the Technical Support at <https://customer.ansys.com>.

5. Tools Version Compatibility

For complete information on the external tools and environments and on the versions supported by the different modules installed with SCADE products, consult *Appendix B - SCADE Products Interoperability with External Tools* in *SCADE Installation Guidelines*.

SCADE Display Release Notes

The following sections present the new features and enhancements for SCADE Display 2025 R2 release.

1. [Fixes and Enhancements](#)
2. [Certification Credits](#)
3. [Limitations and Workarounds](#)
4. [Migration Information](#)
5. [Tools Version Compatibility](#)

1. Fixes and Enhancements

This release of SCADE Display is a maintenance release containing fixes and enhancements.

SCADE Display-Scade One Coupling

- Ability to export graphical specifications to run co-simulation in Scade One
 - New configuration to generate C code, DLLs, and a Scade One project containing an operator that bears the graphical specification's I/Os
 - Co-simulation through a test harness in Scade One
 - Embeddable Scade One and SCADE Display application onto a target with manual integration code

SCADE Test Integration

For details, consult [SCADE Test Release Notes \(p. 271\)](#).

SCADE LifeCycle Integration

For details, consult [SCADE LifeCycle Release Notes \(p. 279\)](#).

2. Certification Credits

The SCADE Display 2025 R2 release includes the following tools with certification credits.

- **SCADE Display KCG Code Generator 6.7.2:** for qualification under DO-178C/DO-330 up to TQL-1 and for certification under ISO 26262, IEC 61508 up to SIL 3, or EN 50128 up to SIL 3/4
- **SCADE Display KCG Code Generator 6.7.1:** for qualification under DO-178C/DO-330 up to TQL-1 and for certification under ISO 26262, IEC 61508 up to SIL 3, or EN 50128 up to SIL 3/4

- See also [Certification Credits \(p. 271\)](#) for SCADE Test and [Certification Credits \(p. 279\)](#) for SCADE LifeCycle

3. Limitations and Workarounds

This section describes the limitations and workaround solutions applying to the SCADE Display 2025 R2 release.

Limitations in SCADE Display Modeling

- Effect containers are incompatible with glTF objects and should not be used with them
- No support of animations internal to glTF objects when using GltfImporter widget
- Masks on active areas are not supported

Limitations in Code Generation for FMUs

- When using FMU states, any modification of model requires to save new FMU states
- When using FMU states, user imported function with memory must not store pointers

Limitation in SCADE Display Simulation

- “Restart” command and “Reset” scenario command in reference object simulation: array and structure variables are not updated to the Variable Dictionary's initial value. Instead, they are initialized to 0

Limitations in SCADE Display KCG & OGLX

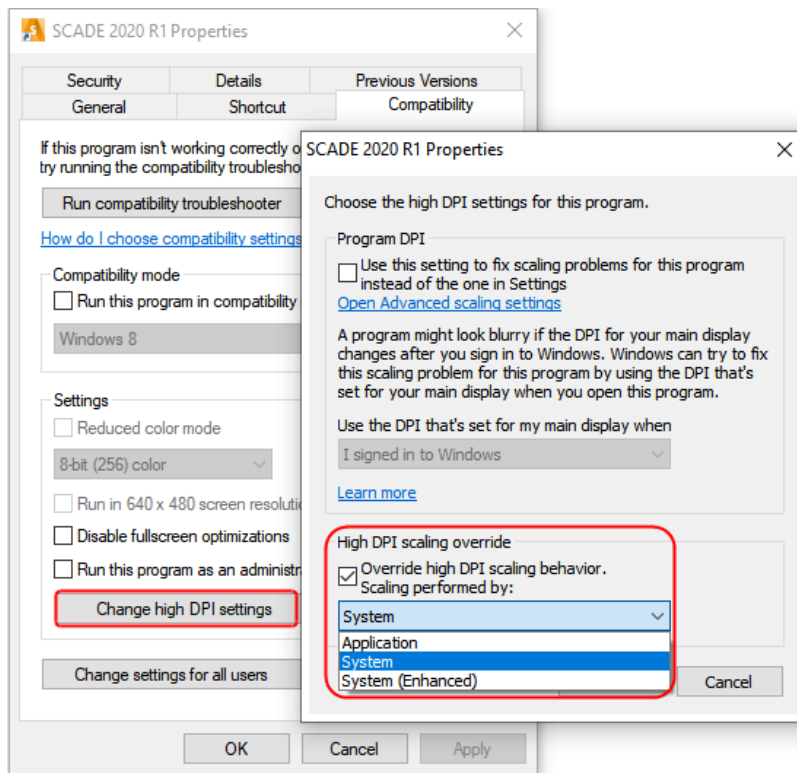
- When drawing a primitive with arc segments, and setting transparency on the outline part of the primitive, darker points appear where the elementary lines used to draw the segment are superposed.
- Clip-planes are not supported in OGLX with OpenGL driver SC 1.0.1.
- Only 16 nested containers with Push/Pop matrix supported with OpenGL driver SC 1.0.1.

Limitation in SCADE Applications

- SCADE applications may not scale well (with symptoms like tiny icons and UI elements) on high-DPI (high-dots-per-inch) displays, depending on the display scaling factor selected for the user's Windows display settings.

Workaround*: users can let Windows automatically manage the scaling of SCADE applications. This may result in applications being blurry, but scaled correctly.

1. Right-click the SCADE product binary (scade.exe) in File Explorer (or Desktop shortcut) to open Properties. On Compatibility tab, click **Change high DPI settings**, under High DPI scaling override section, check the **Override high DPI scaling behavior** option.
2. Select **System** for the **Scaling performed by** drop-down.



* This is only applicable to Windows 10 version “1703” (also called “Creators Update”, delivered in April 2017) and subsequent ones, and can be applied to all SCADE versions. The same operation can be applied to the vcs.exe binary in the same folder.

Limitation in PDF User Documentation

- Hyperlinks in PDF documents opened in some web browsers may not work, preventing navigation between documents.

Workaround: set Adobe Acrobat as default PDF reader.

4. Migration Information

SCADE Display 2025 R1 to 2025 R2

- Format change: at loading, automatic migration of .etp project files

SCADE Display 2024 R2 to 2025 R1

- No format change: no project migration issue
- For projects created before 2025 R1 and loaded in 2025 R1, whenever editing existing Code Generation configuration settings, the configuration information in ETP file is updated as KCG671 for @SDY:CODEGENERATOR property

SCADE Display 2024 R1 to 2024 R2

- No format change: no project migration issue

SCADE Display 2023 R2 to 2024 R1

- Format change: at loading, automatic migration of .etp project files

For migration information related to previous versions, contact the Technical Support at <https://customer.ansys.com>.

5. Tools Version Compatibility

For complete information on the external tools and environments and on the versions supported by the different modules installed with SCADE products, consult *Appendix B - SCADE Products Interoperability with External Tools* in *SCADE Installation Guidelines*.

SCADE Test Release Notes

The following sections present the new features and enhancements for SCADE Test 2025 R2 release.

1. [New Features and Enhancements](#)
2. [Certification Credits](#)
3. [Limitations and Workarounds](#)
4. [Migration Information](#)
5. [Tools Version Compatibility](#)

1. New Features and Enhancements

This section describes the new features and enhancements introduced in SCADE Test 2025 R2 release.

SCADE Test UX improvements

- Improved flow to debug application from SCADE Test: start SCADE Simulator from SCADE Test results view with a single click

SCADE Test Tools for SCADE Suite

- New certification credits for all SCADE Test verification tools (see [Certification Credits \(p. 271\)](#))

SCADE LifeCycle Integration

For details, consult [SCADE LifeCycle Release Notes \(p. 279\)](#).

2. Certification Credits

The SCADE Test 2025 R2 release includes the following tool with certification credits:

- **SCADE Test Environment for Host for SCADE Suite - 2025 R2** for qualification under DO-178C/DO-330 TQL-5 and for certification under ISO 26262 TCL3, IEC 61508 T2, or EN 50128 T2
- **SCADE Test Model Coverage for SCADE Suite - 2025 R2** for qualification under DO-178C/DO-330 TQL-4 and for certification under ISO 26262 TCL3, IEC 61508 T2, or EN 50128 T2 (Ada coverage is outside certification scope)
- **SCADE Test Target Execution for SCADE Suite - 2025 R2** ready for qualification under DO-178C/DO-330 TQL-5 and for certification under ISO 26262 TCL3, IEC 61508 T2, or EN 50128 T2
- **SCADE Test Environment for Host for SCADE Display - 2025 R1** for qualification under DO-178C/DO-330 TQL-5 and for certification under ISO 26262 TCL3, IEC 61508 T2, or EN 50128 T2

- **SCADE Test Model Coverage for SCADE Display - 2025 R1** (for **SCADE Display KCG 6.7.2** usage) for qualification under DO-178C/DO-330 TQL-5 and for certification under ISO 26262 TCL3, IEC 61508 T2, or EN 50128 T2 (Ada coverage is outside certification scope)
- **SCADE Test Model Coverage for SCADE Display - 2021 R2**(for **SCADE Display KCG 6.7.1** usage) for qualification under DO-178C/DO-330 TQL-5 and for certification under ISO 26262 TCL3, IEC 61508 T2, or EN 50128 T2 (Ada coverage is outside certification scope)
- **SCADE Test Target Execution for SCADE Display - 2025 R1** for qualification under DO-178C/DO-330 TQL-5 and for certification under ISO 26262 TCL3, IEC 61508 T2, or EN 50128 T2

3. Limitations and Workarounds

This section describes the limitations and workaround solutions applying to the SCADE Test 2025 R2 release.

Limitation in Coverability Analysis

- Internal failure occurs when analyzing a model containing an operator abstracted by mixed list of imported or non-imported operators and where several abstract operators are polymorphic in type and size.

Workaround:

1. Update the model so that an operator abstracts either imported operators or non-imported operators.
2. Update the model so that it does not contain any size polymorphic abstract operator.

Limitations in Rapid Prototyper

- No support of animations internal to glTF objects when using GltfImporter widget
- Masks on active areas are not supported
- No support for SCADE Suite KCG 6.6 Ada
- Code Generation for FMUs:
 - When using FMU states, any modification of model requires to save FMU states
 - When using FMU states, user imported function with memory must not store pointers

Limitation in Test Execution

- Snapshots not supported by test execution on SCADE Display specifications

Limitations in Test Target Execution

- **Test Harness Generation** (C or Ada): assignments and tests on data of external types, and snapshots are not supported
- **Test Harness for RTRT target** (C): for generated test harnesses to be properly interpreted by RTRT, `kcg_char` must be mapped to `unsigned char`

- **Test Harness for VectorCAST target:**

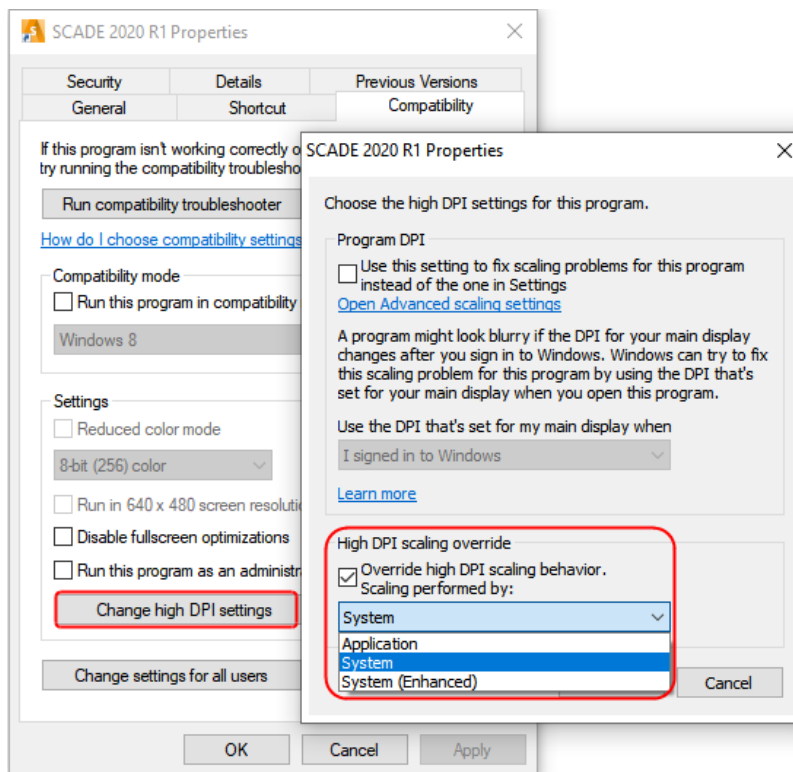
- Models with sensors redefinition cannot be used by VectorCAST test harnesses (C code)
- For generated test harnesses in C code to be properly interpreted by VectorCAST, `kcg_char` must be mapped to `char` and `kcg_true` must be mapped to 1 and `kcg_false` to 0 (default mapping)

Limitation in SCADE Applications

- SCADE applications may not scale well (with symptoms like tiny icons and UI elements) on high-DPI (high-dots-per-inch) displays, depending on the display scaling factor selected for the user's Windows display settings.

Workaround*: users can let Windows automatically manage the scaling of SCADE applications. This may result in applications being blurry, but scaled correctly.

1. Right-click the SCADE product binary (`scade.exe`) in File Explorer (or Desktop shortcut) to open Properties. On Compatibility tab, click **Change high DPI settings**, under High DPI scaling override section, check the **Override high DPI scaling behavior** option.
2. Select **System** for the **Scaling performed by** drop-down.



* This is only applicable to Windows 10 version "1703" (also called "Creators Update," delivered in April 2017) and subsequent ones, and can be applied to all SCADE versions. The same operation can be applied to the `vcs.exe` binary in the same folder.

Limitation in PDF User Documentation

- Hyperlinks in PDF documents opened in some web browsers may not work, preventing navigation between documents.

Workaround: set Adobe Acrobat as default PDF reader.

4. Migration Information

SCADE Test 2025 R1 to 2025 R2

- The installation of SCADE Test Environment for Host for Suite has changed from (SCADE)\SCADE to (SCADE)\SCADE Test\SCTE
- The installation of SCADE Model Coverage for Suite has changed from (SCADE)\SCADE to (SCADE)\SCADE Test\SCMC
- No format change: no project migration issue

SCADE Test 2024 R2 to 2025 R1

- No format change: no project migration issue

SCADE Test 2024 R1 to 2024 R2

- No format change: no project migration issue

SCADE Test 2023 R2 to 2024 R1

- No format change: no project migration issue

SCADE Rapid Prototyper 2025 R1 to 2025 R2

- No format change: no project migration issue

SCADE Rapid Prototyper 2024 R2 to 2025 R1

- No format change: no project migration issue

SCADE Rapid Prototyper 2024 R1 to 2024 R2

- No format change: no project migration issue

SCADE Rapid Prototyper 2023 R2 to 2024 R1

- Format change: at loading, automatic migration of .etp project files

For migration information related to previous versions, contact the Technical Support at <https://customer.ansys.com>.

5. Tools Version Compatibility

For complete information on the external tools and environments and on the versions supported by the different modules installed with SCADE products, consult *Appendix B - SCADE Products Interoperability with External Tools* in *SCADE Installation Guidelines*.

SCADE Architect Release Notes

The following sections present the new features and enhancements for SCADE Architect 2025 R2 release.

1. Fixes and Enhancements
2. Limitations and Workarounds
3. Migration Information
4. Tools Version Compatibility

1. Fixes and Enhancements

This release of SCADE Architect is a maintenance release containing fixes and minor UX Improvements.

SCADE LifeCycle Integration

For details, consult [SCADE LifeCycle Release Notes \(p. 279\)](#).

2. Limitations and Workarounds

This section describes the limitations and workaround solutions applying to the SCADE Architect 2025 R2 release.

Limitations in SCADE Architect Modeling

- Changing type of part/reference not supported
- Turning part into reference not supported
- SCADE Architect diagrams may not display connections and associations properly (for example, alignment of wires shifting) when opening diagram with different DPI (high-dots-per-inch) displays: the option **DPI Rendering** can be used in diagram properties to correct alignment
- Definition of new Guard and Weight properties on existing Control Flows may not display in Activity diagrams: recreate the Control Flow to refresh diagram with new property information.

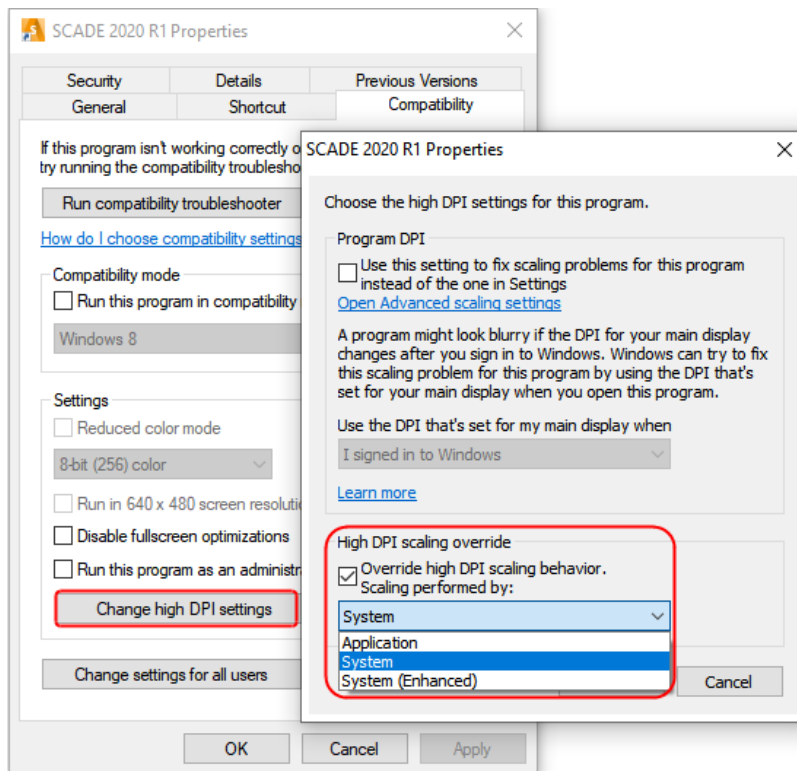
Limitations in SCADE Architect Application

- SCADE Architect may fail to start or may allocate too little memory to its Java Virtual Machine under some rare circumstances (mostly when running in a virtualized environment with machines having less than 8 GB RAM). Default memory allocation can be fine-tuned by adding a specially crafted `sc-sys.ini` file in the `<INSTALLATION>/SCADE/` folder and containing the following line:
`-Xmx1024m`. The value after `-Xmx` is given in megabytes and can be adjusted as required.

- SCADE applications may not scale well (with symptoms like tiny icons and UI elements) on high-DPI (high-dots-per-inch) displays, depending on the display scaling factor selected for the user's Windows display settings.

Workaround*: users can let Windows automatically manage the scaling of SCADE applications. This may result in applications being blurry, but scaled correctly.

1. Right-click the SCADE product binary (scade.exe) in File Explorer (or Desktop shortcut) to open Properties. On Compatibility tab, click **Change high DPI settings**, under High DPI scaling override section, check the **Override high DPI scaling behavior** option.
2. Select **System** for the **Scaling performed by** drop-down.



* This is only applicable to Windows 10 version "1703" (also called "Creators Update", delivered in April 2017) and subsequent ones, and can be applied to all SCADE versions. The same operation can be applied to the vcs.exe binary in the same folder.

Limitations in Model Import/Export

- Table query scripts, ATY files, and content providers must be explicitly added by users when exporting an archive
- Object colors in diagrams are lost when importing Enterprise Architect models

Limitations in Configurator

- Reference establishment is limited to the scope of user metaclasses. It is not possible to directly reference Architect API metaclasses.

- SCADE LifeCycle Reporter ignores configuration aspects and reports configured models as standard SCADE Architect models.
- During a working session, it is required to open only SCADE Architect projects or only SCADE Architect Configurator projects. If ever alternating projects, one must close the current session and reopen the application. A message warns you about potential conflicts or loading errors whenever switching between SCADE Architect projects and SCADE Architect Configurator projects in the same session.

Limitations in Diff/Merge

- Diagrams, Sequences, and Use Cases are only matched/ merged as a whole (no matching/merging of sub-elements).
- Notes with structured types are accessible as a whole from API and merged as a whole.

Limitation in Architect API

- SCADE Architect Java API for annotations depends on ANTLR 3.2.0 (Java component parsing textual files). The API cannot be installed on an Eclipse target platform that includes a more recent and incompatible version of the **org.antlr.runtime** plug-in. Recent versions of the XText Eclipse modeling component rely on ANTLR 4 and cannot be installed on the same platform as the API.
- SCADE Architect API has no support for concepts coming from Use Case or Sequence diagrams.

Limitation in PDF User Documentation

- Hyperlinks in PDF documents opened in some web browsers may not work, preventing navigation between documents.

Workaround: set Adobe Acrobat as default PDF reader.

3. Migration Information

SCADE Architect 2025 R1 to 2025 R2

- No specific issue: see [SCADE Architect \(each release\)](#) (p. 277)

SCADE Architect 2024 R2 to 2025 R1

- No specific issue: see [SCADE Architect \(each release\)](#) (p. 277)

SCADE Architect 2024 R1 to 2024 R2

- No specific issue: see [SCADE Architect \(each release\)](#) (p. 277)

SCADE Architect (each release)

- SCADE Architect projects referencing libraries and/or configurations must be properly migrated. The migration of projects with libraries can be automated by a script distributed within SCADE installation and located at \SCADE\scripts\Migration\Architect.
- If ever using an in-house configuration, it is necessary to regenerate it and redeploy it within teams using it. If the configuration contains libraries, one has to do the same for libraries, starting

from them. For all projects, configured and not, it is necessary to load/save all models, starting from libraries if any.

- Synchronization may not work properly when pushing to a SCADE Architect model that was not first migrated to the latest tool version.

For migration information related to previous versions, contact the Technical Support at <https://customer.ansys.com>.

4. Tools Version Compatibility

For complete information on the external tools and environments and on the versions supported by the different modules installed with SCADE products, consult *Appendix B - SCADE Products Interoperability with External Tools* in *SCADE Installation Guidelines*.

SCADE LifeCycle Release Notes

The following sections present the new features and enhancements for SCADE LifeCycle 2025 R2 release.

1. [New Features and Enhancements](#)
2. [Certification Credits](#)
3. [Limitations and Workarounds](#)
4. [Migration Information](#)
5. [Tools Version Compatibility](#)

1. New Features and Enhancements

This section describes the new features and enhancements introduced in SCADE LifeCycle 2025 R2 release.

SCADE LifeCycle ALM Gateway

- Support of Working Set for Codebeamer™
- New capability to import attributes with Polarion® Connector

SCADE LifeCycle Reporter

- Renewed certification credits for SCADE LifeCycle Reporter for SCADE Suite (see [Certification Credits \(p. 279\)](#))

2. Certification Credits

The SCADE LifeCycle 2025 R2 release includes the following tool with certification credits:

- **SCADE LifeCycle Reporter 2025 R2 for SCADE Suite Certification Data Rev B**
 - For qualification under DO-178C (DO-330 TQL-5)
 - For certification under ISO 26262, IEC 61508, and EN 50128
- **SCADE LifeCycle Reporter 2025 R1 for SCADE Display Certification Data**
 - For qualification under DO-178C (DO-330 TQL-5)
 - For certification under ISO 26262, IEC 61508, and EN 50128
- **SCADE LifeCycle Reporter 2020 R2c for SCADE Test Certification Data**
 - For qualification under DO-178C (DO-330 TQL-5)

- For certification under ISO 26262, IEC 61508, and EN 50128
- **SCADE LifeCycle Reporter 2024 R1 for SCADE ARINC 661 UA DF Generator Certification Data**
 - For qualification under DO-178C (DO-330 TQL-5)
- **SCADE LifeCycle Model Change 2021 R2 for SCADE Suite Certification Data Rev B**
 - For qualification under DO-178C (DO-330 TQL-5)
 - For certification under ISO 26262, IEC 61508, and EN 50128

3. Limitations and Workarounds

This section describes the limitations and workaround solutions applying to the SCADE LifeCycle 2025 R2 release.

Limitations in Requirements Management

- Exporting models to DOORS NG, Jama, or Polarion is slow (API-related issue from each ALM tool).

Workaround: use custom model export from UI to reduce elements to export and launch export in batch.

- **Codebeamer Connections:**

- If API throttling is active in Codebeamer ALM tool, import/export is slower.

- **Polarion Connections:**

- When importing a Polarion document in a baseline, if the document contains both work items and headings at the same level, they might be ordered differently than in Polarion because of equal outline numbers.
 - Filtered documents cannot be imported if work items are referenced from another document (Polarion API issue)
 - Impossible to delete exported elements: Polarion API adds a "TO_DELETE" tag on such elements.

Workaround: filter elements tagged by "TO_DELETE".

- **DOORS NG Connections:**

- Selecting views is not available in DOORS Next Generation 6.0.5 & 6.0.6 connection, if Jazz Team Server has no Global Configuration Manager installed.
 - When working with DOORS NG 6.0.4, accessing the local configuration of DOORS NG is limited to the content of the initial stream.

Workaround: fixed by using 6.0.5 or higher.

- **DOORS 9.x (native) Connections:**

- When using DOORS baselines for traceability, requirement location by navigation from SCADE projects to ALM tools is not possible.
- **SCADE Suite:**
 - It is not possible to trace model objects defined textually in .scade files. [CR110-44]
 - Exporting model objects defined textually in .scade files causes object duplication or errors in DOORS, DOORS NG, Polarion, or Jama tools.
- **SCADE Architect:**
 - When using default full export, it is not possible to export back to ALM tool databases an AUTOSAR, Avionics, FACE, or any SCADE Architect configured model. Instead use custom model export.

Limitation in Reporter

- **SCADE Architect:** configured models are reported as standard SCADE Architect models and configuration aspects are ignored.
- **SCADE Suite, SCADE Architect, and SCADE Test:** Use of Unicode UTF-8 option in Windows Regional settings is not supported for report generation.

Limitation in PDF User Documentation

- Hyperlinks in PDF documents opened in some web browsers may not work, preventing navigation between documents.

Workaround: set Adobe Acrobat as default PDF reader.

4. Migration Information

ALM Gateway 2025 R1 to 2025 R2

- No format change: no project migration issue

ALM Gateway 2024 R2 to 2025 R1

- No format change: no project migration issue

ALM Gateway 2024 R1 to 2024 R2

- When exporting models with images, all images (not exported previously due to a bug) are exported at once for the first export using the new release.

ALM Gateway 2023 R2 to 2024 R1

- No format change: no project migration issue

For migration information related to previous versions, contact the Technical Support at <https://customer.ansys.com>.

5. Tools Version Compatibility

For complete information on the external tools and environments and on the versions supported by the different modules installed with SCADE products, consult *Appendix B - SCADE Products Interoperability with External Tools* in *SCADE Installation Guidelines*.

SCADE Solutions for ARINC 661 Compliant Systems

Release Notes

The following sections present the new features and enhancements for SCADE 2025 R2 Solutions for ARINC 661 release.

1. [New Features and Enhancements](#)
2. [Certification Credits](#)
3. [Limitations and Workarounds](#)
4. [Migration Information](#)

1. New Features and Enhancements

This section describes the new features and enhancements introduced in SCADE 2025 R2 Solutions for ARINC 661 release.

Error and Logging UX Improvements

- Rationalize error messages throughout the ARINC 661 Solution to ease development (SCADE server, SCADE UA Page Creator, and SRVGEN tool):
 - Centralize error collection/logging mechanisms
 - Carry error location information all the way

SCADE ARINC 661 Server

- Server debug traces
 - Initialization phase: diagnose Definition File parsing issues
 - Runtime phase: diagnose communication/concurrency issues
 - Controlled via environment variable `SERVER_DEBUG`

SCADE LifeCycle Integration

- For details, consult [SCADE LifeCycle Release Notes \(p. 279\)](#).

2. Certification Credits

The SCADE 2025 R2 Solutions for ARINC 661 release includes the following tool with certification credits:

- **SCADE UA DF Generator 6.8**: for qualification under DO-178C/DO-330 up to TQL-1

- See also [Certification Credits \(p. 279\)](#) for SCADE LifeCycle Reporter for SCADE ARINC 661 UA DF Generator

3. Limitations and Workarounds

This section describes the limitations and workaround solutions applying to the SCADE 2025 R2 Solutions for ARINC 661 release.

Limitation in A661 Server Compiled with ANGLE

- On a laptop computer with Intel integrated graphics (driver version 27.20.100.8336 or upper), the server may freeze after few minutes if it is run on this graphics chip. Please contact Ansys SCADE Support in case this issue occurs.

Limitation in SCADE UA Page Creator

- The following widgets have no preview for display in Look Definition Preview: ActiveArea, CursorOver, MapHorz_ItemList, MapVert_ItemList, MapHorz_Container, SizeToFitContainer, ShuffleToFitContainer, TabbedPanel

Limitations in A661 Server

The solution is compatible with Supplement 9 of the ARINC 661 standard, with the following limitations:

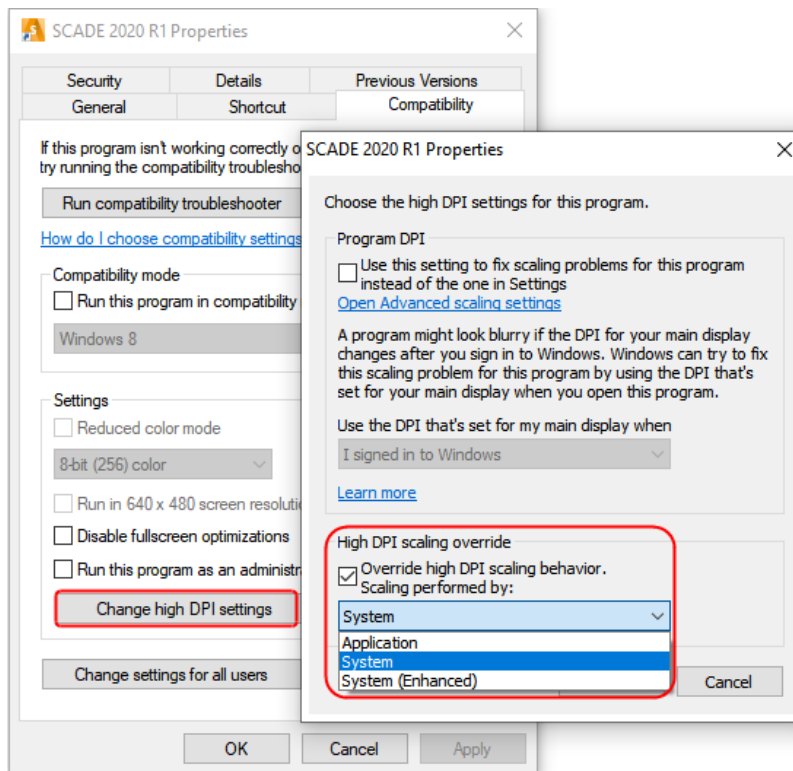
- **Non-supported widgets:** CameraAbsolute3D, CameraLookAt3D, FramingRefreshContainer, GpTriangleFan, GpTriangleStrip, GpVertexBuffer, GpVertexRender, LayoutHorizontal, LayoutVertical, LightAmbient3D, LightDistant3D, LightList3D, LightPoint3D, LightSpot3D, Map3D, Map3D_ItemList, Map3D_Source, Map3D_VertexBuffer, pExternalSource, PointerRestrictionArea, PointerWall, ScaleContainer, Selector, Tree
- **Non-supported extensions:** BoundaryCheckExtension, ColumnExtension, CursorMapEventsExtension, DefaultEntryExtension, DefaultTextExtension, EditTextOpenEventExtension, ExcludedRegionsExtension, HorizontalScrollExtension, MapHorzCoordSystemEventExtension, MapVertCoordSystemEventExtension, PopupPriorityExtension, ReorderExtension, StaticParamBufferExtension, StyleSetExtension, StyleSetParameterExtension, TimeoutExtension, VerticalScrollExtension
- **Non-supported symbol commands:** SetScale, SymbolReference
- Some widgets, widget extensions, or symbol commands may have limitations that are listed in the corresponding SRS document
- Animation escape sequences are not supported in Complex Texts (LabelComplex, ScrollList widgets)
- Animation laws defined in Definition Files are not supported
- The A661_REQ_DEBUG_WIDGETS debug widget request and the A661_NOTE_DEBUG_WIDGETS notification are not supported
- SubPictures are not supported

Limitation in SCADE Applications

- SCADE applications may not scale well (with symptoms like tiny icons and UI elements) on high-DPI (high-dots-per-inch) displays, depending on the display scaling factor selected for the user's Windows display settings.

Workaround*: users can let Windows automatically manage the scaling of SCADE applications. This may result in applications being blurry, but scaled correctly.

1. Right-click the SCADE product binary (scade.exe) in File Explorer (or Desktop shortcut) to open Properties. On Compatibility tab, click **Change high DPI settings**, under High DPI scaling override section, check the **Override high DPI scaling behavior** option.
2. Select **System** for the **Scaling performed by** drop-down.



* This is only applicable to Windows 10 version "1703" (also called "Creators Update", delivered in April 2017) and subsequent ones, and can be applied to all SCADE versions. The same operation can be applied to the vcs.exe binary in the same folder.

Limitation in PDF User Documentation

- Hyperlinks in PDF documents opened in some web browsers may not work, preventing navigation between documents.

Workaround: set Adobe Acrobat as default PDF reader.

4. Migration Information

SCADE 2025 R1 to 2025 R2 Solutions for ARINC 661

- Format change: at loading, automatic migration of UAPC projects, Server Creator projects, and SCADE Display projects used to design widgets

SCADE 2024 R2 to 2025 R1 Solutions for ARINC 661

- No format change: no project migration issue

SCADE 2024 R1 to 2024 R2 Solutions for ARINC 661

- UAPC models using the default A661 Configuration files: automatic migration from A661-6 to A661-9 library format upon loading. For some widgets having modified parameters, their value is lost and the default value is associated with the new parameter:
 - CheckButton: parameter PicturePosition is renamed as CheckboxPosition and is set to A661_LEFT
 - SymbolAnimated: LoopType renamed as LoopFlag and is set to A661_LOOP_FORWARD

For Constants: various categories renamed, constant type is set to uncategorized

SCADE 2023 R2 to 2024 R1 Solutions for ARINC 661

- Format change: at loading, automatic migration of UAPC projects
- Server Creator projects: automatic split of widget capacities/definitions into separate files

For migration information related to previous versions, contact the Technical Support at <https://customer.ansys.com>.

SCADE Avionics Package Release Notes

The following sections present the new features and enhancements for SCADE Avionics Package 2025 R2 release.

- 1. Fixes and Enhancements
- 2. Limitations and Workarounds
- 3. Migration Information

1. Fixes and Enhancements

This release of SCADE Avionics Package is a maintenance release containing fixes and enhancements..

FACE Modeling

- New possibility to consider “unbounded string” and translate it to bounded string:

2. Limitations and Workarounds

This section describes the limitations and workaround solutions applying to the SCADE Avionics Package 2025 R2 release.

Limitations in SCADE Avionics Package

- Due to CTS known issue, the char "0" should not be used in package naming. CTS checks fail and complain about missing .h/.hpp files if ever using this character in package naming.

Workaround: use CHK_394 to detect and fix invalid naming of concerned elements.

- **FACE 3.0 Template language:** lowerbound and upperbound are not interpreted for the generation of the IDL Sequence type when their value are different. BoundedSequence is supported, but not unbounded Sequence. The following constructs are not supported: @inline annotation, @optional annotation, generic templates.

Limitation in PDF User Documentation

- Hyperlinks in PDF documents opened in some web browsers may not work, preventing navigation between documents.

Workaround: set Adobe Acrobat as default PDF reader.

3. Migration Information

SCADE Avionics Package 2025 R1 to 2025 R2

- No specific issue: see [SCADE Avionics Package \(each release\) \(p. 288\)](#)

SCADE Avionics Package 2024 R2 to 2025 R1

- No specific issue: see [SCADE Avionics Package \(each release\) \(p. 288\)](#)

SCADE Avionics Package 2024 R1 to 2024 R2

- No specific issue: see [SCADE Avionics Package \(each release\) \(p. 288\)](#)

SCADE Avionics Package 2020 Rx to All Releases

- Existing FACE projects using the SDM delivered as part of SCADE product installation are migrated automatically at project load.
- When using another SDM, the SDM plugin for SCADE must be migrated as follows:
 1. **SDM Migration:** Copy the SDM plugin in a new directory. Then, the SCADE SDM model contained in that directory must be migrated; this is done simply by loading/saving the model with the new SCADE release.
 2. **SDM reference from FACE projects:** using any text editor, in each ETP project file, update the "@SYSTEM:DROPINS" property to the new SDM directory. In the following example, verify the last reference to sdm plugin corresponds to the migrated SDM.

```
<Prop id="55" name="@SYSTEM:DROPINS">
  <value>$(SCADE)/dropins/face</value>
  <value>$(USER_PLUGINS_FOLDER)</value>
  <value>$(SCADE)/dropins/face/sdm/3.1.4</value>
</Prop>
```

Warning: a project must not reference several SDM plugins. Note: to avoid updating this property, put the SDM directories under the \$(SCADE) variable that expands into the installation path of the current SCADE release as shown in example above. Alternatively, use any user environment variable.

3. **FACE projects can then be migrated in the same way as all SCADE Architect projects:** as explained for each release migration.

SCADE Avionics Package (each release)

- SCADE Architect projects referencing libraries and/or configurations must be properly migrated. The migration of projects with libraries can be automated by a script distributed within SCADE installation and located at \SCADE\scripts\Migration\Architect. For more information, check ReadMe available here and section about "Managing Project Compatibility and Migration" in SCADE Architect User Manual.
- If ever using an in-house configuration extending the FACE or AADL (or Avionics) configuration provided with SCADE Avionics Package, it is necessary to regenerate the in-house configuration and redeploy it within teams using it. If the configuration contains libraries, one has to do the same for libraries, starting from them.
- If using configured projects (based on configurations provided with the product or based on in-house configuration previously updated), it is necessary to load/save all models, starting from libraries if any.

For migration information related to previous versions, contact the Technical Support at <https://customer.ansys.com>.

