

Release Notes

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

Release 2026 R1
March 2026

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

Copyright and Trademark Information

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

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

Disclaimer Notice

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

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

U.S. Government Rights

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

Third-Party Software

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

Published in the U.S.A.

Table of Contents

Global xvii
1. Advisories xvii
2. Compatibility with Previous Releases xix
3. Installation xix
4. Licensing xx
5. Documentation xxii
6. Verification Manuals xxii
6.1. Workbench Verification Manual xxii
6.1.1. New Workbench Test Cases xxii
6.1.2. Modified Workbench Test Cases xxii
6.1.3. Removed Workbench Test Cases xxii
6.2. Mechanical APDL Verification Manual xxiii
6.2.1. New Mechanical APDL Test Cases xxiii
6.2.2. Modified Mechanical APDL Test Cases xxiii
6.2.3. Removed Mechanical APDL Test Cases xxiii
6.3. Ansys LS-DYNA Verification Manual xxiii
6.3.1. New LS-DYNA Test Cases xxiii
6.3.2. Modified LS-DYNA Test Cases xxvi
6.3.3. Removed LS-DYNA Test Cases xxvii
7. Online Video Access xxvii
8. Ansys Customer Site xxvii
Ansys 3D Design Products 1
Discovery Application Release Notes 3
1. Discovery Application Product Advisories 3
2. Discovery Application Enhancements Overview 3
2.1. Fluid Flow Simulation 4
2.1.1. Compressible Flow 4
2.1.2. Fluids Solver 4
2.1.3. Capture Sharp Edges 5
2.1.4. Capture Thin Structures 6
2.1.5. Geometry Discretization Warnings 6
2.2. Contacts and Interfaces 6
2.2.1. Fluid-to-Fluid Interfaces and Virtual Walls 6
2.2.2. Thermal and Electric Conductance Definition 7
2.2.3. Electric Conditions and Joule Heating 8
2.3. Transfer to the Icepak Application 8
2.3.1. Transfer Fluid, Thermal or Fluid-thermal conditions to Icepak application 8
2.4. Transfer to the Mechanical Application 9
2.4.1. Transfer Fluid-Solid (Conjugate) Heat Transfer Data to Mechanical Application 9
2.5. Solution and Optimization 9
2.5.1. Defined Variations Sampling Algorithm 9
2.5.2. Sensitivity Analysis and Total Coefficient of Prognosis 9
2.6. Modeling Tools 9
2.6.1. Improvements to Modeling Documentation 10
2.6.2. Sketch Tool Improvements 10
2.6.3. Bend Tool 10
2.6.4. Combine Tool 10
2.6.5. Interference Tool 11
2.6.6. Suppression Tool 11

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

2.6.7. Detect Tool 11
2.6.8. Edit Faceted Bodies 11
2.7. Query Editor 12
2.8. History Tracking 12
2.9. Materials 13
2.10. User Interface Enhancements 13
2.10.1. Default UI Themes 13
2.10.2. Beams Assignment 13
2.10.3. Extension Builder 13
2.10.4. Export to SimAI 13
2.10.5. Import Granta Cloud Materials 14
2.10.6. Welcome Page 14
2.11. User Settings Summary 14
2.11.1. General 14
2.11.2. Licensing 14
2.11.3. Physics 14
2.11.3.1. Fluids Solver 14
2.11.3.2. Capture Sharp Edges 15
2.11.3.3. Capture Thin Structures 15
2.11.3.4. Fluid-to-Fluid Interfaces 16
2.11.4. Results 16
2.11.4.1. Location Setting for Fluid Flow Monitors 16
2.11.4.2. Saved Scenes Remote Download Image Resolution 16
2.11.4.3. Include Explore Results 16
2.11.5. Advanced 17
2.11.6. Cloud Services 17
2.12. Results 17
2.12.1. Viscous and Pressure Force Monitors 17
2.12.2. Wall Yplus Monitors 18
2.12.3. Results Comparison Dashboard 18
2.12.4. Location Setting for Fluid Flow Monitors 18
2.12.5. Tabular Results 18
2.12.6. Structural Chart Results 19
2.12.7. Filtered Seeding 20
2.12.8. Particle Count 21
2.13. Sample Models 21
2.14. Performance 22
2.15. Installation and Licensing 22
2.15.1. Installer Enhancements 22
2.15.2. License Preference Overrides 23
2.15.3. On Demand License Level Change 23
2.16. Ansys 3D Design Verification Manual 23
2.17. Ansys Discovery Technology Showcase: Example Problems 23
3. Discovery Application Caveats and Known Issues 23
SpaceClaim Release Notes 25
1. SpaceClaim Product Advisories 25
2. SpaceClaim Changes Overview 25
2.1. Installation 25
2.2. Scripting 26
2.3. Import/Export Support 26
3. SpaceClaim Caveats and Known Issues 26

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

DesignModeler Release Notes 29
1. DesignModeler Product Advisories 29
2. DesignModeler Enhancements 29
3. DesignModeler Caveats and Known Issues 29
Ansys Structural Products 31
Mechanical Application Release Notes 33
1. Changes in Product Behavior 33
2. Documentation 36
3. Performance 37
4. Add-ons 37
5. Graphics 39
6. Geometry 39
7. External Model 40
8. Composites 40
9. Contact and Connections 40
10. Tables 41
11. Mesh 41
12. Additive Manufacturing Process Simulation 41
13. Analysis 42
14. Fracture 43
15. Substructure Analysis 43
16. Structural Optimization 44
17. Motion 44
18. Imported Loading Conditions 45
19. Solution 46
20. Explicit Dynamics 47
21. Results 50
22. Scripting 51
Mechanical APDL 53
1. Structural 53
1.1. Contact 53
1.1.1. Enhancement of GCGEN and GCDEF commands for better performance and reliability 54
1.1.2. Moment Convergence Check for Pilot nodes for Structures with No Other Rotational DOFs 54
1.1.3. Automatic Merging of Overlapping Multi-point Constraint (MPC) Contact Pairs 54
1.1.4. New Command Stabilizes Nonlinear Solutions 54
1.1.5. CNTGEN and SFGEN Commands Now Support Nodal Components 54
1.1.6. Performance Improvements for Multiple Spot Welds 55
1.1.7. 2D-3D Bonded MPC Contact for Solid Assemblies Supports Thermal and Structure-thermal Coupling 55
1.1.8. DMP Performance Improvement of MPC CONTA175 Element 55
1.1.9. Improved Convergence when Simulating Assemblies that Include Gasket Elements 55
1.2. Elements and Nonlinear Technology 56
1.2.1. Contact-based Displacement Increment Control 56
1.2.2. Updates to Smart Stop Criterion 56
1.2.3. Gasket Element Improvement 56
1.2.4. Direct Element Embedding Enhancement 56
1.2.5. General Axisymmetric Element Enhancements 57
1.2.6. Updates to 3D Structural Shell Elements 57
1.2.7. Extended support for Nonlocal Damage Formulation 57

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

1.2.8. New Standard Variant for Void Creation in Generalized Remeshing for Adaptive Nonlinear Damage (GRAND) 57
1.2.9. Automatic Identification and Removal of Orphaned Element Patches in Nonlinear Adaptivity 57
1.2.10. Support for Arruda-Boyce, Bergstrom-Boyce and TNM Material Models in Nonlinear Adaptivity 57
1.3. Material and Fracture Modeling 58
1.3.1. Finite-strain Material Models Enhancement 58
1.3.2. Material Curve-Fitting Enhancements 58
1.3.3. Support for Anisothermal Derivatives in Thermal Analyses 58
1.3.4. Damage Cap Enhancements 59
1.3.5. Improved UserCreep Handling in the Extended Drucker-Prager Cap Creep Model 59
1.3.6. Advanced Grain-Growth Model for Sintering 59
1.3.7. New Sintering Stress Evolution Models 59
1.3.8. Conversion of Initial Stress to Crack-surface Traction in Multiple Loadsteps 59
1.3.9. Enhancement of SIF (Stress-Intensity Factors) Calculation 59
1.3.10. SMART Support with Substructuring 59
1.3.11. New Fracture Benchmark Problem 60
1.3.12. Support of Mixed Element Types in SMART Crack-Growth Analysis 60
1.4. Linear Dynamics 60
1.4.1. Enhanced Restart Capability for Harmonic Balance Method Analysis 60
1.4.2. Multistage Cyclic Symmetry Supports Transient Thermal Analysis 60
1.4.3. Joints with Cyclic Symmetry Linear Perturbation Harmonic Analyses 60
2. Multiphysics 60
2.1. Acoustics 61
2.1.1. Aerodynamic Source Terms (Lighthill's Analogy) in the Wave Equation 61
2.1.2. Perfectly Matched Layer (PML) for Axially Symmetric Elements in Transient Analysis 61
2.1.3. Logarithmic absorption function option for Perfectly Matched Layer (PML) and Irregular Perfectly Matched Layer (IPML) acoustic elements 61
2.2. Thermal 61
2.2.1. New NMISC Values for Performing Energy Balance 61
2.2.2. View Factor Sums for Radiation Surface Facets 62
2.3. Coupled-Field 62
2.3.1. Expanded Capability of User-Programmable Features 62
2.3.2. **SHELL229 Supports Structural-thermal and Structural-thermoelectric Analyses** 62
2.3.3. Extended Support for Nonlocal Damage Formulation 62
3. Solvers 63
3.1. Resource Prediction Feature 63
3.2. PCG Block Lanczos Eigensolver 63
3.3. PCG Solver Enhancements 63
4. High-Performance Computing 64
4.1. Distributed-Memory Parallel Processing Enhancements 64
4.2. GPU Acceleration Feature: New Support and Updates 64
4.3. Recommended Devices for the GPU Accelerator: New and Updated GPUs 64
5. Other Enhancements 65
5.1. Compiler Update for User-Programmable Features on Linux Systems 65
5.2. New Coded Database File Command 66
5.3. Modified Coded Database File Command 66
5.4. Expanded Element Face Component Creation Options 66
6. Commands 66

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

6.1. New Commands 66
6.2. Modified Commands 67
6.3. Undocumented Commands 68
7. Elements 68
7.1. New Elements 68
7.2. Modified Elements 69
7.3. Undocumented Elements 70
8. Documentation 70
8.1. Cyclic Symmetry Documentation Examples 70
8.2. Updates for Programmers 70
8.3. *Feature Archive* 70
9. Technology Showcase: Example Problems 70
10. Mechanical APDL Release 2026 R1 Update Guide 71
10.1. Backward Compatibility 71
10.2. Feature Updates Causing Result or Behavior Changes 71
10.2.1. Change in `.ldhi` Files 72
10.2.2. Changes in Automatic Hybrid Parallel 72
10.2.3. Change in Internal Multipoint Constraint (MPC) Reporting in .ldhi and .cnd Files 73
10.2.4. Moment Convergence Check at Pilot Node of Target Elements with Rotational DOFs 73
10.2.5. Harmonic Balance Method Uses Sparse Solver by Default 73
10.2.6. Output Displays Elapsed Time 73
10.2.7. Change in Format of Nonlinear Solution Monitor File (`.mntr`) 74
10.2.8. Restart Mechanism Improves Conjugate Gradient (CG) Solver Robustness 74
10.2.9. Change in Preconditioned Conjugate Gradient (PCG) Lanczos Eigensolver 74
10.2.10. Output behavior of mapdl.run in a *PYTHON block 74
10.2.11. **Change in Impedance Sheet Formulation for Acoustic Analysis** 75
10.2.12. Change in Default for Ansys Product Improvement Program (APIP) 75
10.2.13. Change in Behavior Due to Backstress Scaling in the Chaboche Model for Non-Isothermal Loading 75
10.3. Known Incompatibilities 75
10.4. Known Issues 75
10.5. Resolved Issues and Defects 76
10.6. Features Removed or Archived 77
10.6.1. Undocumented Real Constant and Keyoption for Contact Elements 77
10.6.2. Connection Commands 77
Autodyn 79
1. New Features and Enhancements 79
Aqwa 81
1. Aqwa Solver Modules 81
2. Aqwa Graphical Supervisor (AGS) 81
3. Hydrodynamic Analysis Systems 81
Ansys Composite Prep (ACP) Release Notes 85
1. New Features and Enhancements 85
1.1. Interface to the Ansys LS-DYNA™ Application 85
1.2. Solid Composite Model with Reinforcing Elements 86
1.3. Performance Improvements 86
1.4. Split Modeling Ply into Production Materials 86
1.5. End of Support for `.acp` Serialization Format 86
1.6. Sampling Point for Solid Models 87
1.7. Improved Search Widget 87

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

1.8. Ansys Composite Cure Simulation (ACCS) Updates 87
1.9. Two Examples of Solid Modeling with Reinforcing Elements 87
2. Known Limitations 87
Material Designer Release Notes 89
1. Notes 89
2. Known Limitations 89
Additive Manufacturing 91
1. PBF Process Simulations 91
2. DED Process Simulations 93
3. Sintering Process Simulations 94
4. Distortion Compensation 95
5. Changes in Product Behavior 95
6. AM-Related Mechanical APDL Commands 96
7. PyAdditive 96
Sherlock 99
1. Known Issues 99
2. New Features 99
3. Resolved Issues and Limitations 99
4. Scheduled Updates 99
5. General Enhancements 100
5.1. Scoring Updates 100
5.2. Updates to Sherlock Component Models 100
5.3. Updates to Sherlock Analysis Modules 101
5.4. Solder Updates 102
5.5. Updates to Ansys Workbench/Mechanical Application Integration 102
5.6. GUI Updates 102
5.7. Performance Improvements 103
5.8. Other Enhancements 103
6. API Updates 103
6.1. General API Improvements 103
6.2. New APIs 104
6.3. Modified APIs 104
6.4. Deprecated APIs 105
6.5. Previously Deprecated APIs Removed 105
7. Bug Fixes 105
8. Third Party Dependency Changes 107
Release Notes 109
1. Geometry & Preprocessing 109
1.1. Sketch Simple Curves Directly in Ansys Forming 109
1.2. Edit Curves for Faster, More Flexible Design 110
1.3. Guided Pin Generation for Greater Flexibility in Tool Design 111
1.4. Transition to Parasolid Geometric Kernel 111
2. Forming Process Enhancements 112
2.1. Improved Support for Solid Elements 112
2.2. Tipping Support for One Step Projects 113
2.3. Double Drawbead Now Supported 113
2.4. Define Multiple Offset Values for Local Tooling Regions 114
2.5. Flexible Gravity Simulation 114
2.6. Operation Suspension in Forming Workflows 115
3. Solver & Performance Improvements 115
3.1. Enhanced Implicit Solver Performance 115

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

3.2. Improved Speed at Binder Closing 115
4. Postprocessing & Reporting 115
4.1. Automatic Report Generation 116
4.2. New Advanced Solution Type: Stochastic Simulation 116
4.3. Edge to Drawbead Function 117
4.4. New Post-Processing Variable: Maximum FLC Failure 118
4.5. Support for Formability Index with Barlat 2000 118
5. Documentation 118
5.1. New Forming Keyword Reference Manual 118
Motion Release Notes 119
1. New Features 119
2. Functional Enhancements 119
LS-DYNA Release Notes 121
1. New Documents 121
2. LS-PrePost Enhancements 121
2.1. Geometry 122
2.2. Preprocessing 122
2.3. Postprocessing 123
2.4. Application 124
2.5. Solution Explorer 125
2.6. Post Explorer 126
2.7. General 126
3. LS-Run Enhancements 127
4. Envyo Application 127
4.1. ACP-to-DYNA Mapping Option Enhancements 127
4.2. New Workflow Examples 128
4.3. Performance Improvements 128
Ansys Fluids Products 129
Fluent 131
1. Supported Platforms for Ansys Fluent 2026 R1 131
2. New Features in Ansys Fluent 2026 R1 131
2.1. Meshing Mode 131
2.2. Solution Mode 132
2.3. Fluent Workspace Applications 144
3. Updates Affecting Ansys Fluent 2026 R1 Code Behavior 147
3.1. Meshing Mode 148
3.2. Solution Mode 148
3.3. Fluent Workspace Applications 150
4. Features To Be Removed in a Future Release 151
CFX 153
1. Supported Platforms 153
2. New Features and Enhancements 153
3. Incompatibilities 154
CFD-Post 155
1. Supported Platforms 155
2. New Features and Enhancements 155
TurboSystem Release Notes 157
1. Supported Platforms 157
2. New Features and Enhancements 157
TurboGrid 159
1. Supported Platforms 159

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

2. New Features and Enhancements 159
BladeBuilder 161
1. Supported Platforms 161
2. New Features and Enhancements 161
Polyflow Classic 163
1. New Features 163
2. Supported Platforms 163
3. Resolved Issues and Limitations in Ansys Polyflow Classic 2026 R1 163
Forte 165
1. New Features and Enhancements 165
2. Resolved Issues and Limitations 165
Chemkin 167
1. New Features and Enhancements 167
2. Resolved Issues and Limitations 168
FENSAP-ICE 169
1. New Features and Enhancements in Ansys FENSAP-ICE 169
2. Resolved Issues and Limitations 169
EnSight 171
1. New Features and Enhancements in Ansys EnSight 171
2. PyEnSight 172
3. Ansys Dynamic Reporting (Nexus) 172
4. Beta in This Release 173
5. Alpha in This Release 173
6. Resolved Issues and Limitations 173
FreeFlow Release Notes 175
1. New Features 175
1.1. SPH Adaptive Sizing (Beta) 175
1.2. General New Features (SPH) 176
1.3. General New Features (Solver SDK) 176
1.4. Ansys FreeFlow Coupling with Ansys optiSLang Software Feature 176
2. Improvements 176
2.1. Performance Improvements 177
2.2. User Experience Improvements 177
2.3. Postprocessing Improvements 177
3. FreeFlow Modules 177
4. FreeFlow Scripts 178
5. Beta in This Release 178
6. Resolved Issues and Limitations 179
Ansys Rocky Release Notes 181
1. New Features 181
1.1. General New Features (DEM) 181
1.2. General New Features (SPH) 181
1.3. General New Features (Rocky SDK) 182
1.4. Ansys Rocky Coupling with Ansys Fluent Software Features 182
1.5. Ansys Rocky Coupling with Ansys optiSLang Software Feature 182
2. Improvements 182
2.1. PrePost Scripting Improvements 182
2.2. Performance Improvements 182
2.3. User Experience Improvements 183
2.4. Postprocessing Improvements 183
3. Rocky Modules 183

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

4. Rocky Scripts 184
5. Beta in This Release 184
6. Ansys Engineering Copilot within Ansys Rocky (Beta) 185
7. Resolved Issues and Limitations 186
Thermal Desktop 2026 R1 Release Notes 187
1. New features and capabilities 187
Ansys Electronics Products 189
Icepak 191
1. Introduction 191
2. New and Modified Features in Ansys Icepak 2026 R1 191
3. Resolved Issues and Limitations in Ansys Icepak 2026 R1 191
Ansys Optical Products 193
Speos Release Notes 195
1. Features from BETA to Official 195
2. Productivity Enhancement 195
3. Sensor / Autonomous Driving 199
4. Result Experience 200
5. Optical Part Design 201
6. Changes 203
7. Resolved Issues and Limitations 204
Speos for NX Release Notes 205
1. Productivity Enhancement 205
2. Sensor / Autonomous Driving 208
3. Result Experience 208
4. Optical Part Design 209
5. Changes 210
6. Resolved Issues and Limitations 211
Speos for Creo Parametric Release Notes 213
1. Productivity Enhancement 213
2. Result Experience 215
3. Changes 217
4. Resolved Issues and Limitations 218
Speos Labs Release Notes 219
1. New Features and Enhancements 219
2. Changes 220
3. Resolved Issues and Limitations 221
Speos HPC Release Notes 223
1. New Features and Enhancements 223
2. Resolved Issues and Limitations 223
Ansys Acoustics Products 225
Ansys Sound Release Notes 227
1. Changes in Product Behavior 227
2. Ansys Sound: Analysis and Specification 227
2.1. New Features 227
2.2. Enhancements 228
2.3. Resolved Issues and Limitations 228
3. Ansys Sound: Jury Listening Test 228
3.1. Enhancements 228
3.2. Resolved Issues and Limitations 228
Ansys Autonomous Vehicle Products 229
AVxcelerate Lighting and Sensors Release Notes 231

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

1. Enhancement 231
1.1. Co-simulation with CarMaker 14 231
1.2. Enhanced security added to gRPC communication 231
2. Changes 231
2.1. Integrated Plugins 231
2.2. Updated Installation Paths 231
2.3. Windows 10 No Longer Supported 232
AVxcelerate Sensors Release Notes 233
1. New Features 233
1.1. OSI Input Availability added to AVX Simulations 233
1.2. IP Protection for Camera Sensors 233
2. Enhancements 234
2.1. Camera Simulation with Light Propagation Engine 234
2.1.1. New Raytraced Ambient Occlusion 234
2.1.2. New Raytraced Reflections 234
2.1.3. Improved Raytraced Shadows 235
2.1.4. **Enhancement to the Fog feature** 235
2.1.5. **Rain in simulation** 235
2.2. Thermal Camera Sensor Creation and Edition 236
2.3. GPU Direct Memory Access (DDMA) to Sensors Outputs 237
2.4. Co-simulation with CarMaker 14 237
2.5. Radar Simulation Improvement with Respect to Curved Surfaces 237
2.6. New Tracks Added 237
2.7. Addition of NCAP 2026 Assets and Track 238
2.8. Enhanced security added to gRPC communication used within the AVxcelerate Sensors™ solution 238
2.8.1. AVxcelerate Sensors Simulator and API 238
2.8.2. Asset Preparation API 238
3. Changes 239
3.1. AVxcelerate Sensors Simulator API v1 239
3.2. Discontinued Tools and Samples 239
3.3. AVSimulation SCANeR™ Connector 239
3.4. Windows 10 No Longer Supported 239
Ansys Mesh Prep Products 241
Meshing 243
1. Changes in Product Behavior from Previous Releases 243
2. Topology Suppress 243
3. Morph Control 244
4. Enhancements to Mesh Workflows 244
5. Enhancements to MultiZone Method 244
6. Enhancements to Weld 244
7. Enhancements to Meshing Options 245
8. Enhancements to Sizing Group 245
ICEM CFD 247
1. Import Geometry Options 247
Ansys Connect Products 249
Ansys Minerva 251
1. New Features and Enhancements 251
1.1. User Experience 251
1.2. Data 252
1.3. Configuration 252

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

2. Resolved Issues 253
3. Known Issues and Limitations 253
4. Documentation Enhancements 254
Distributed Compute Gateway (DCG) 255
1. New Features and Enhancements 255
2. Resolved Issues 255
3. Known Issues and Limitations 255
optiSLang 257
1. General 257
2. User Interface 259
3. Connectors 259
4. Algorithms 262
ModelCenter Release Notes 265
1. ModelCenter Desktop 265
2. ModelCenter Remote Execution 265
3. ModelCenter MBSE 265
Ansys Platform Products 267
ACT 269
Ansys Dynamic Reporting 271
Ansys Viewer 273
1. Advisory: Ansys Viewer End of Life 273
2. New Features and Enhancements 273
3. Resolved Issues and Limitations 273
4. Known Issues and Limitations 273
CAD Release Notes 275
Distributed Compute Services (DCS) 277
DesignXplorer 279
Remote Solve Manager (RSM) 281
1. New Features and Enhancements 281
2. Issues Resolved in this Release 281
3. Known Issues and Limitations 281
System Coupling 283
1. Participants 283
2. Mapping Accuracy 284
3. System Coupling in Workbench 284
4. Mapping Analysis Type 284
5. System Coupling Infrastructure 284
6. Data Model 284
7. Participant Library 285
8. Known Issues and Limitations 285
Workbench 287
1. General Workbench Enhancements 287
2. Engineering Data Workspace 287
3. Injection Molding Data 288
Embedded Software Products 289
Scade One Release Notes 291
1. New Features and Enhancements 291
2. Limitations/Known Issues 293
SCADE Suite Release Notes 297
1. New Features and Enhancements 297
2. Certification Credits 298

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

3. Limitations and Workarounds 298
4. Migration Information 300
5. Tools Version Compatibility 300
SCADE Display Release Notes 301
1. Fixes and Enhancements 301
2. Certification Credits 301
3. Limitations and Workarounds 302
4. Migration Information 303
5. Tools Version Compatibility 303
SCADE Test Release Notes 305
1. New Features and Enhancements 305
2. Certification Credits 306
3. Limitations and Workarounds 306
4. Migration Information 308
5. Tools Version Compatibility 309
SCADE Architect Release Notes 311
1. Fixes and Enhancements 311
2. Limitations and Workarounds 311
3. Migration Information 313
4. Tools Version Compatibility 314
SCADE LifeCycle Release Notes 315
1. New Features and Enhancements 315
2. Certification Credits 315
3. Limitations and Workarounds 316
4. Migration Information 317
5. Tools Version Compatibility 317
SCADE Solutions for ARINC 661 Compliant Systems Release Notes 319
1. New Features and Enhancements 319
2. Certification Credits 320
3. Limitations and Workarounds 320
4. Migration Information 322
SCADE Avionics Package Release Notes 323
1. Fixes and Enhancements 323
2. Limitations and Workarounds 323
3. Migration Information 324

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

List of Figures

1. Sketch Interface with Sample Spline 110
2. Curve Editing Interface with Sample Curve Split into Four Segments 111
3. FLD for a Model with Solid Elements 113
4. Sample Project with Double Drawbeads Defined 114
5. Sample Slide Generated with Automatic Report 116
6. Sample Analysis in LS-OPT 117
7. Example of an Edge to Drawbead Draw-In Plot 118
8. Example Volume Rendering with Mesh 140
9. Example Inline Unit Drop-down 143

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

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

Global Release Notes

The release notes are specific to Ansys, Inc. Release 2026 R1 and arranged by application or product, with the exception of:

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

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 2026 R1 can be found in the online version of 2025 R2 release notes . Or via ansys.com> Customers> Customer Center> General Support> Download Center> Downloads> Product Documentation> Select Release> PDF packages> release.

1. Advisories

- Release 2026 R1 will be the final version of the Distributed Compute Gateway (DCG) application. Support for DCG will end in 2027.

  The functionality of DCG 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.

  For more information, see the **Ansys HPC Platform Services** page on the Ansys Help site or contact Ansys Support.

- 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.

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

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 was the final version of the Ansys Additive desktop application, which includes the Additive Print and Additive Science modules. No further updates or bug fixes are provided.

  As of Ansys release 2025 R2, the Ansys Additive application was removed from the Ansys Installation program, and support for it ended.

  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 is no longer available.

  The functionality of DCS has been transitioned to Ansys HPC Platform Services — the next-generation solution for HPC workflow management from Ansys.

  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. xxvii) 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 2026 R1, 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.

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

2. Compatibility with Previous Releases

Backwards Compatibility: Ansys 2026 R1 was tested to read and resume databases from the following previous versions: 2024 R1, 2024 R2, 2025 R1, and 2025 R2. 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 2026 R1 first resume it in a supported version and then resume that database in 2026 R1.

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 2026 R1. Review these items carefully.

- For this release, the Ansys installation program has added the following products:
  - 3D Design
    - → Headless Geometry Service (PyAnsysGeometry)
  - AI Tools
    - → Actionable Language Interface (ALI)
    - → Embedding Model BGE-M3
  - Connect
    - → Behavior Execution Engine
    - → System Architecture Modeler (SAM)
    - → optiSLang Core Headless
    - → optiSLang Core UI
  - Electronics
    - → Motor-CAD
    - → PerceiveEM
  - Embedded Software
    - → Scade One
    - → SCADE Rapid Prototyper
  - Fluids
    - → RBF Morph Fluids

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

  - Structures

    → RBF Morph Structures

- Need for PAC file, and its URL, are automatically detected from operating system settings.

4. Licensing

The following enhancements were made to Ansys, Inc. Licensing for Release 2026 R1:

Desktop:

- The Ansys License Manager has been updated to FlexNet Publisher client version 11.19.9.0.
- The Ansys License Manager has been updated to OpenSSL version: 3.5.4.
- The Ansys License Manager has been updated to Java version: 21.0.8.9.
- The Ansys License Manager has been updated to Tomcat version: 10.1.48.
- The Ansys License Manager has been updated to Curlt version: 8.16.0.
- Added support for IPv6 to Elastic Licensing.
- General security and performance improvements.
- The ANSYSLI_ prefix has been changed toANSYS_LICENSING_.
- UI Changes:
  - The left-hand menu has been restructured.
  - A single location to opt in or out of Ansys Product Improvement Program (APIP) for all applications.
  - A Windows-specific proxy option removed on Linux.

Web:

- Web Licensing performance improvements:
  - Faster license checkouts
  - Expanded health checks and failover mechanisms
  - Resiliency against third-party Cloud infrastructure and service issues
- Web Licensing Assignment improvements
  - Automatic Assignment updates when entitlements expire
- Shared Web Denial reporting improvements

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

  - Transactions table now shows best Product match if customer does not have a product with the requested feature(s)
  - Related transactions are now grouped for easier interpretation
  - Denials are now available in charts
- Added access to the following Products, Missions and Bundles:
  - Missions AI+
  - SNOPT optimizer for spacecraft trajectory design
  - STK
    - → Engine Deployment
    - → Enterprise
    - → Premium Air
    - → Premium Space
    - → Pro
    - → RF Channel Modeler Enterprise
    - → RF Channel Modeler Radar
    - → Scheduler
    - → Shield Plus
  - Digital Missions Bundle
  - Structures Explorer
  - GRANTA Advanced Materials
  - GRANTA MI
  - GRANTA MI Editor
  - GRANTA MI Viewer
  - GRANTA Selector

Licensing Portal and ID Portal:

- Enhancements to the Licensing Portal include:
  - Page header “breadcrumbs” now clearly indicate which page is active
  - Convenient buttons to download table data, clear all filters, and reset the view

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

- New option to remove existing Members before importing from a file (Available for both Users and Groups)

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 2026 R1:

6.1. Workbench Verification Manual

6.2. Mechanical APDL Verification Manual

6.3. Ansys LS-DYNA Verification Manual

6.1. Workbench Verification Manual

6.1.1. New Workbench Test Cases

No test cases were added for 2026 R1.

6.1.2. Modified Workbench Test Cases

No test cases were modified for 2026 R1.

6.1.3. Removed Workbench Test Cases

No test cases were removed for 2026 R1.

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

6.2. Mechanical APDL Verification Manual

6.2.1. New Mechanical APDL Test Cases

- VM324: **Swelling Strain of a Block Under High Fluence Loading:** A fluence is applied on a block constrained in the vertical direction. This induces a swelling strain. The value of the swelling strain is compared to Mechanical APDL results.

6.2.2. Modified Mechanical APDL Test Cases

- VM174: **Bimetallic Beam Under Thermal Load:** 4-Node Coupled Field Shell (SHELL229) added to the test case.
- VM223: **Electro-Thermal Microactuator Analysis:** 4-Node Coupled Field Shell (SHELL229) added to the test case.

6.2.3. Removed Mechanical APDL Test Cases

No test cases were removed for 2026 R1.

6.3. Ansys LS-DYNA Verification Manual

Updates for 2026 R1:

6.3.1. New LS-DYNA Test Cases
6.3.2. Modified LS-DYNA Test Cases
6.3.3. Removed LS-DYNA Test Cases

6.3.1. New LS-DYNA Test Cases

LS-DYNA Workbench Test Cases

No test cases were added for 2026 R1.

LS-DYNA Solver Test Cases

- VM-LSDYNA-SOLVE-038: **Rectangular Plate with Circular Hole Subjected to Tensile Loading:** The simulation models a 3D rectangular plate with a central circular hole, a geometric irregularity, subjected to linear static structural analysis to evaluate stress concentration effects.
- VM-LSDYNA-SOLVE-039: **Buckling of a Uniform Rod:** This test case models the elastic buckling behavior of a uniform cantilever rod under axial compressive loading.
- VM-LSDYNA-SOLVE-040: **Buckling of a Stepped Rod:** This finite element (FE) simulation investigates the elastic buckling behavior of a stepped cantilever rod subjected to two axial loads: a tensile force and a compressive force.
- VM-LSDYNA-SOLVE-041: **Buckling of a Curved Panel:** This finite element (FE) simulation investigates the elastic buckling behavior of a curved panel under a uniform radial pressure.

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

- VM-LSDYNA-SOLVE-042: **Modal Analysis of an Annular Plate:** This test case models the free vibration of an annular plate, subjected to a free-clamped condition. The objective is to validate the first six natural frequencies of the structure.
- VM-LSDYNA-SOLVE-043: **Fundamental Frequency of an Elliptical Plate**: This test case simulates the vibration of a clamped-edge elliptical plate to validate its fundamental natural frequency.
- VM-LSDYNA-SOLVE-044: **Modal Analysis of a Rectangular Plate:** This simulation models the vibration of a rectangular plate with the two smaller edges subjected to a simply supported condition and one of the other edges subjected to a clamped condition. The objective is to validate the first six natural frequencies of the structure.
- VM-LSDYNA-SOLVE-045: **Fundamental Frequency of a Square Plate with Opening:** This simulation investigates the vibration of a clamped squared plate with a central opening.
- VM-LSDYNA-SOLVE-046: **Fundamental Frequency of a Slender Beam with Central Mass:** This test case models the vibration of a pinned-pinned slender bar with a concentrated mass at its mid-length. The objective is to validate the fundamental natural frequency of the structure.
- VM-LSDYNA-SOLVE-047: **Axial Natural Frequency of Hat Beams:** This test case simulates the vibration of a spring-mass system composed of two collinear hat bars. The objective is to validate the axial natural frequency of the structure.
- VM-LSDYNA-SOLVE-048: **Transverse Natural Frequencies of a Cantilever Tube:** This test case models the vibration of a cantilever tube.
- VM-LSDYNA-SOLVE-049: **Natural Frequency of a Spring-Mass System:** This test case models models a spring-mass system comprised of an instrument set on a rubber mount. The objective is to validate the system's natural frequency.
- VM-LSDYNA-SOLVE-050: **Lateral Vibration of a Rectangular Plate:** This test case models the lateral vibration of a rectangular plate with one smaller edge subjected to a clamped condition. The objective is to validate the structure's first natural frequency of lateral vibration.
- VM-LSDYNA-SOLVE-051: **Natural Frequencies of a Circular Plate:** This test case models the vibration of a circular plate with its edge subjected to a clamped condition.
- VM-LSDYNA-SOLVE-052: **Natural Frequencies of a Two-Mass-Spring System:** This test case models a system composed of two masses and three springs. The objective is to validate the two normal modes and natural frequencies of the system.
- VM-LSDYNA-SOLVE-053: **Seismic Response of a Mass-Spring System:** This test case models a vibrometer, consisting of a mass-spring system with base excitation.
- VM-LSDYNA-SOLVE-054: **Seismic Response of a Mass-Spring-Damper System - Equivalent Structural Damping:** This test case models a mass-spring-damper system with a harmonic disturbing force acting on the mass.
- VM-LSDYNA-SOLVE-055: **Transient Response of a Mass-Spring-Damper System:** This test case models a mass-spring-damper system, which is displaced by a distance Δ when released. Four damping ratios are tested. The objective is to validate the displacement profile of each system.

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

- VM-LSDYNA-SOLVE-056: **Transient Response of a Two-Mass-Spring System:** This test case models a system composed of two masses and two springs. Mass 1 is subjected to a pulse load F. The objective is to validate the displacement profile of each mass.
- VM-LSDYNA-SOLVE-057: **Pin-Ended Double Cross: In-Plane Vibration:** This test case models eight beams of the same length, evenly spaced, and connected at the center. All endpoints are constrained in the x and y directions. The objective is to validate the frequency of predicted modes.
- VM-LSDYNA-SOLVE-058: **Buckling of a Bar with Hinged Ends (Beam Elements):** This test case models the elastic buckling of a long, uniform bar with hinged ends subjected to axial loading. The objective is to validate the critical buckling load of the bar.
- VM-LSDYNA-SOLVE-059: **Portal Frame Under Symmetric Loading:** This test case models a rectangular frame subjected to a uniform distributed load.
- VM-LSDYNA-SOLVE-060: **Seismic Response of a Simply Supported Beam:** This test case models a simply-supported beam with both supports subjected to vertical motion. The objective is to validate the natural frequency, the maximum deflection, and the maximum bending stress of the beam.
- VM-LSDYNA-SOLVE-061: **Transient Response of a Ball Impacting a Flexible Surface:** This test case models a rigid ball dropped onto a flexible surface. The objective is to validate the time, displacement, velocity, and kinetic energy of the ball at the moment of the impact and the maximum displacement.
- VM-LSDYNA-SOLVE-062: **Cantilever with Off-Center Point Masses:** A cantilever beam is situated horizontally with two off-center lump masses. The beam is constrained in all DOFs at the left end. Frequency vibration analysis is performed on the model.
- VM-LSDYNA-SOLVE-063: **Cylindrical Vessel Under Internal Pressure:** This test case models a long cylindrical vessel with closed ends, subjected to an internal pressure. The 3D cylinder is modeled as a 2D axisymmetric rectangle representing the cylindrical shell, and the objective is to validate the circumferential stress of the vessel.
- VM-LSDYNA-SOLVE-064: **Deep Simply-Supported Beam:** This test case performs frequency vibration analysis on a simply-supported beam that is constrained in the x, y, z and Rx direction on the left endpoint and in the y and z direction on the right endpoint.

LS-DYNA Fluids Test Cases

- VM-LSDYNA-FLUID-008: **2D Passive Scalar Transport in a Laminar Channel Flow:** This test case models the transport of a passive scalar (species) in a two-dimensional, incompressible, steady, laminar flow inside a channel.
- VM-LSDYNA-FLUID-009: **Womersley Flow: Pulsative Flow in a Rigid Cylindrical Tube:** This test case models the Womersley flow which consists of an oscillatory, fully developed laminar flow of a Newtonian fluid in a rigid cylindrical tube driven by a time-periodic pressure gradient.
- VM-LSDYNA-FLUID-010: **Conical Body Impact on a Water Free-Surface:** This test case models the impact of a conical body on a water free-surface. The objective is to validate the vertical hydrodynamic force profile over time.

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

- VM-LSDYNA-SPH-001: **2D Laminar Couette Flow Simulation (SPH):** The simulation uses SPH to model classical Couette flow, representing a viscous, incompressible fluid confined between two parallel plates.

LS-DYNA EMAG Test Cases

No test cases were added for 2026 R1.

LS-DYNA Benchmark Test Cases

No test cases were added for 2026 R1.

LS-DYNA Impact Test Cases

- VM-LSDYNA-IMPACT-001: **Normal Collision of a Rubber Sphere with a Steel Plate:** This test case models the normal collision of a deformable rubber sphere with a rigid steel plate.
- VM-LSDYNA-IMPACT-002: **Tensile Elastic Wave Analysis in a 3D Split-Hopkinson Pressure Bar:** This test case models the three-dimensional impact of a hollow cylinder (striker) on a flange connected to a long bar (incident bar).
- VM-LSDYNA-IMPACT-003: **Tensile Elastic Wave Analysis in a 2D Split-Hopinson Pressure Bar:** This test case models the two-dimensional impact of a hollow cylinder (striker) on a flange connected to a long bar (incident bar).
- VM-LSDYNA-IMPACT-004: **Compressive Elastic Wave Analysis in a 3D Split-Hopkinson Pressure Bar:** This test case models the three-dimensional impact between two long cylinders (a moving striker and stationary incident bar).
- VM-LSDYNA-IMPACT-005: **Compressive Elastic Wave Analysis in a 2D Split-Hopkinson Pressure Bar:** This test case models the two-dimensional impact between two long cylinders (a moving striker and stationary incident bar).

LS-DYNA Dynamics Test Cases

- VM-LSDYNA-DYNAMICS-001: **Uniform Precession of Gyroscope (Rigid Parts):** This test case calculates the precession angular speed of a gyroscope to validate the LS-DYNA dynamics calculations for spinning bodies, specifically rotational motion. All components are modeled as rigid bodies.
- VM-LSDYNA-DYNAMICS-002: **Uniform Precession of Gyroscope (Elastic Solid Disk):** This test case calculates the precession angular speed of a gyroscope (gyro) to validate the LS-DYNA dynamics calculations for spinning bodies, particularly rotational motion. All components, except for the elastic disk (flywheel), are modeled as rigid bodies.
- VM-LSDYNA-DYNAMICS-003: **Uniform Precession of Gyroscope (Elastic Shell Disk):** This test case calculates the precession angular speed of a gyroscope (gyro) to validate the LS-DYNA dynamics calculations for spinning bodies, particularly rotational motion. All components, except for the elastic disk (flywheel), are modeled as rigid bodies.

6.3.2. Modified LS-DYNA Test Cases

No test cases have been modified for 2026 R1.

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

6.3.3. Removed LS-DYNA Test Cases

No test cases have been removed for 2026 R1.

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.

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

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

Ansys 3D Design Products

Release Notes

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

Discovery (p. 3)
SpaceClaim (p. 25)
DesignModeler (p. 29)

Discovery Application Release Notes

Release 2026 R1 of the Discovery application contains the following.

1. Discovery Application Product Advisories
2. Discovery Application Enhancements Overview
3. Discovery Application Caveats and Known Issues

1. Discovery Application Product Advisories

Be aware of the advisories and product changes described here.

Installation and Licensing Advisories

- Operating System Support
  - Support for Microsoft Windows 10 (all versions) has been dropped.
  - Support for Microsoft Windows 11 version 22H2 has been dropped.
- The Discovery application's CUDA Toolkit version is now updated to 12.8 GA. Synopsys recommends the latest drivers, with the minimum required driver version for minor version compatibility being >=528.33.
- Discovery application 2025 R2 was the final version that included support for NVIDIA Kepler-based and Maxwell-based graphics cards. Starting with Release 2026 R1, the application only supports 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.

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. 3) 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 Application Product Advisories (p. 3).

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

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

2.1. Fluid Flow Simulation
2.2. Contacts and Interfaces
2.3. Transfer to the Icepak Application
2.4. Transfer to the Mechanical Application
2.5. Solution and Optimization
2.6. Modeling Tools
2.7. Query Editor
2.8. History Tracking
2.9. Materials
2.10. User Interface Enhancements
2.11. User Settings Summary
2.12. Results
2.13. Sample Models
2.14. Performance
2.15. Installation and Licensing
2.16. Ansys 3D Design Verification Manual
2.17. Ansys Discovery Technology Showcase: Example Problems

2.1. Fluid Flow Simulation

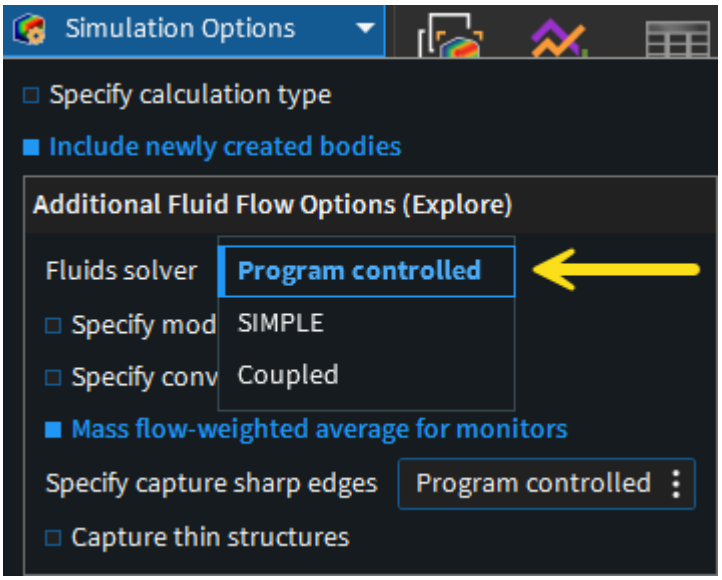
2.1.1. Compressible Flow

Compressible flow now supports a velocity inlet. In past releases, the application only supported a mass flow or a pressure inlet.

2.1.2. Fluids Solver

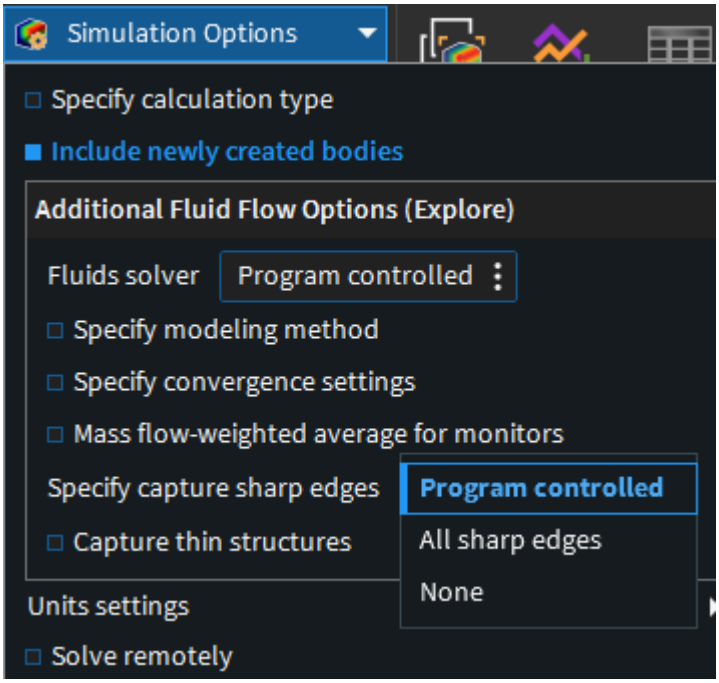
Fluid flow simulations now support additional solver settings in **Explore** and **Refine**. From the **Simulation Options** dropdown menu on the **Simulation** tab, under **Additional Fluid Flow Options**, the default **Fluids solver** is set to **Program Controlled**. This setting allows the application to determine the most appropriate pressure-based solver for the simulation. Choose **SIMPLE** (a segregated algorithm) for steady-state incompressible simulations, or **Coupled** for steady-state and time-dependent compressible flows, as well as flows with a strong swirling component.

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



2.1.3. Capture Sharp Edges

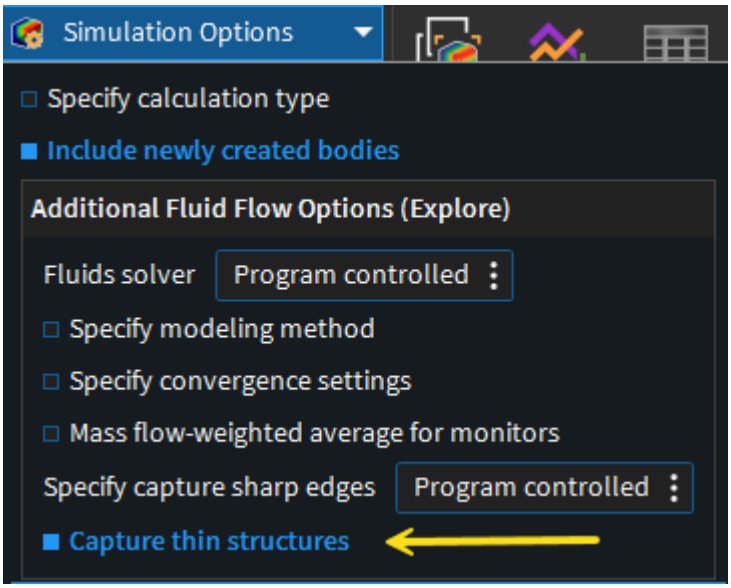
The capture sharp edges option in **Explore** is enhanced with additional options for more flexibility in capturing sharp edges on your model. From the **Simulation Options** dropdown menu in the **Simulation** tab ribbon, under **Additional Fluid Flow Options (Explore)**, the default setting for **Specify capture sharp edges** is **Program Controlled**. This default option allows the application to choose how to best capture sharp edges on your model. You can change the setting to capture **All sharp edges** instead, and the application will attempt to capture all sharp edges in the mesh leading to increased accuracy. However, choosing this setting can increase computation time and decrease solution stability. Otherwise, choose **None** to avoid capturing sharp edges.



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

2.1.4. Capture Thin Structures

Capture thin structures is a new fluids meshing option added to improve discretization of thin regions and accurately represent mesh areas with thicknesses smaller than the local size. With this feature, the application creates stretched (anisotropic) cells along thin directions, which can help lower the cell count compared to isotropic refinement. If this option is not enabled, isotropic cells are generated and very thin regions are merged or overlooked, possibly leading to a loss of detail.



2.1.5. Geometry Discretization Warnings

If the current fidelity level is insufficient when solving a fluid flow simulation in **Explore**, the application generates a warning whenever the discretized body volume used by the solver differs by more than 25% from the geometry body volume. Additionally, a warning is issued if the discretized face area used by the solver differs by more than 25% from the geometry face area for all physics conditions. These warnings ensure that included cutter bodies, where the solid region is not suppressed, are effectively managed. Note that this is not applicable to excluded cutter bodies.

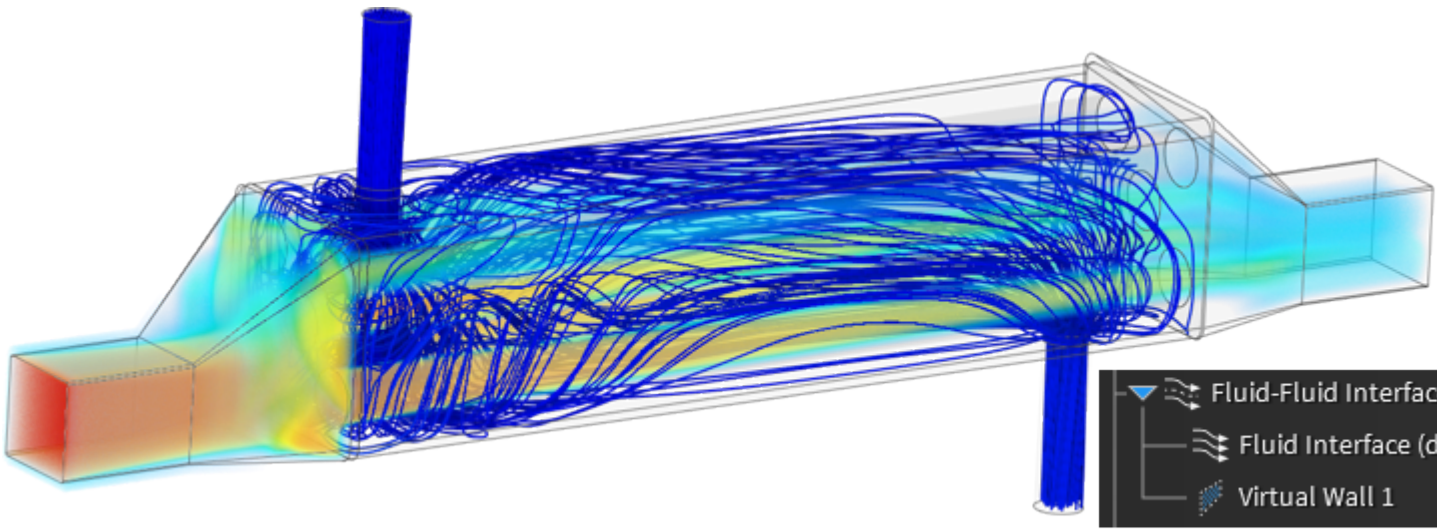
2.2. Contacts and Interfaces

2.2.1. Fluid-to-Fluid Interfaces and Virtual Walls

Under **File Menu > Settings > Physics > Workflow Options**, **Show fluid-to-fluid interfaces** is now selected by default. Fluid interfaces for simulations with fluid-to-fluid interfaces will be visible in the physics tree.

You can now specify a virtual wall between two fluid regions and choose whether to define the virtual wall to be a perfect conductor or a perfect insulator.

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

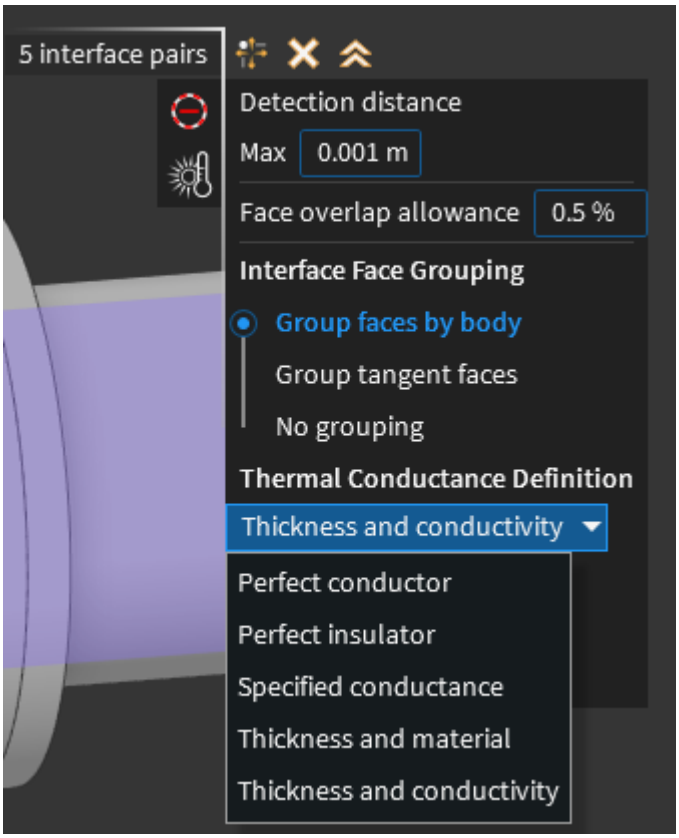


2.2.2. Thermal and Electric Conductance Definition

Contacting surfaces and fluid-solid interfaces now include additional methods for defining thermal and electrical conductance. You can now choose from the following methods:

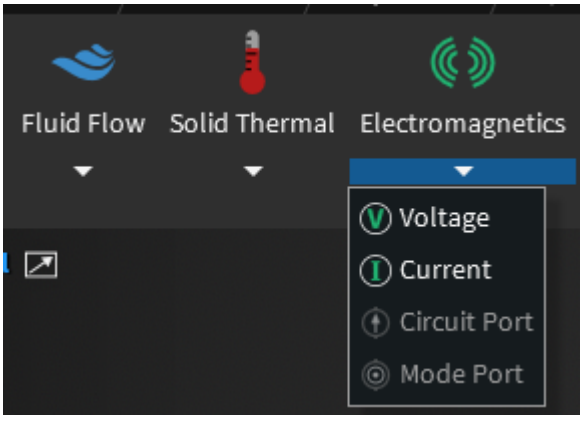
- Perfect conductor
- Perfect insulator
- User specified value for conductance
- Material and thickness-based conductance
- Thickness and conductivity-based conductance

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



2.2.3. Electric Conditions and Joule Heating

Fluid-solid heat transfer simulations now support electric conditions and allow you to model Joule heating, as well as enabling the free or forced convection simulation of busbars, fuses, conductors, electronic traces, etc.



2.3. Transfer to the Icepak Application

2.3.1. Transfer Fluid, Thermal or Fluid-thermal conditions to Icepak application

Discovery application data can now be transferred to the **Icepak** application. If fluid, thermal, fluid-thermal, or fluid-thermal with electric conduction conditions exist, the **Icepak** application will be launched with **.aedti** files.

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

2.4. Transfer to the Mechanical Application

2.4.1. Transfer Fluid-Solid (Conjugate) Heat Transfer Data to Mechanical Application

You can now transfer fluid-solid (conjugate) heat transfer simulations from a completed Discovery simulation to the **Mechanical** application using one of the following options:

- Structural only - transfers structural analysis data, including static pressure results at the fluid-solid interface and solid body temperature results.
- Thermal only - transfers thermal analysis data, such as heat transfer coefficients and near-wall temperature results at the fluid-solid interface, along with solid thermal or electric thermal conditions.
- Linked Thermal Structural - transfers thermal analysis data, including heat transfer coefficients and near-wall temperature results at the fluid-solid interface, as well as solid thermal or electric thermal conditions. Structural analysis data, including static pressure at the fluid-solid interface and a one-way transfer of solid body temperature from the thermal analysis, is also included.

2.5. Solution and Optimization

2.5.1. Defined Variations Sampling Algorithm

When generating variations using the sampling algorithm, define the target number of variations by entering a value. The target number entry box replaces the previous slider.

2.5.2. Sensitivity Analysis and Total Coefficient of Prognosis

When working with defined variations, the Sensitivity analysis investigates the influence of parameterized inputs on outputs. The application displays the analysis results as a predictive approximation quality and sensitivity matrix (Coefficient of Prognosis matrix) and visualizes the functional relationship between two inputs and a selected output (response) as a 3D interactive response surface plot. Together, the matrix and response surface plot assist you in deciding whether to continue with the current approximation model or whether additional sampling data or inputs are needed for improvement.

2.6. Modeling Tools

Note:

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

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

2.6.1. Improvements to Modeling Documentation

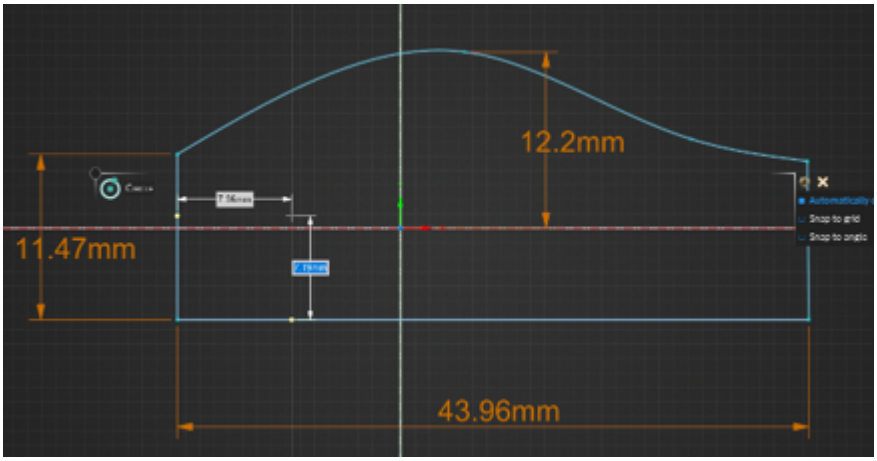
Additional modeling documentation originally developed for the SpaceClaim application is now available in the Discovery Documentation's Table of Contents. This change provides easier access to detailed conceptual and reference information about modeling functionality.

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

2.6.2. Sketch Tool Improvements

The sketch tool includes the following:

- New automatic dimension support - Explicitly specifying a dimension while sketching will automatically generate a dimension constraint.
- Spline knots now support dimension constraints.



- Improved display of dimensions and constraints.

2.6.3. Bend Tool

The **Bend** tool is now available in the Discovery application.

You can bend straight lines and edges to form an arc. You can also adjust the radius of arcs and arced edges.

2.6.4. Combine Tool

The **Combine** tool now includes the following new options:

- The **Combine tolerance** option enables you to combine bodies which cannot normally be combined with the **Combine** tool. Set an appropriate **Combine tolerance** in the tool to merge or subtract bodies which cannot be merged otherwise or produce small slivers which will be eliminated.

  This was previously available as a beta feature.

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

- The **Transfer named selections** option enables you to transfer named selections while merging or subtracting bodies using the **Combine** tool.

2.6.5. Interference Tool

The **Interference tolerance** option enables you to detect and remove interference between bodies that cannot normally be removed with the **Interference** tool. Set an appropriate **Interference tolerance** in the tool to remove the interference between the bodies.

2.6.6. Suppression Tool

The **Suppression** tool enables you to suppress edges or vertices in your model.

- As an alternative method for cleaning up your model, you can mark edges or vertices as suppressed so that they will not be considered in downstream applications such as the Mechanical application. For example, you can suppress edges or vertices if they may result in problematic mesh elements or extra refinement.
- The repair tools **Split Edges**, **Extra Edges**, and **Small Faces** include an additional option to suppress topologies. Extra edges or vertices in the model can be marked as suppressed using this option.
- While in the **Suppression** tool, the application highlights the suppressed edges or vertices to indicate them in the display.

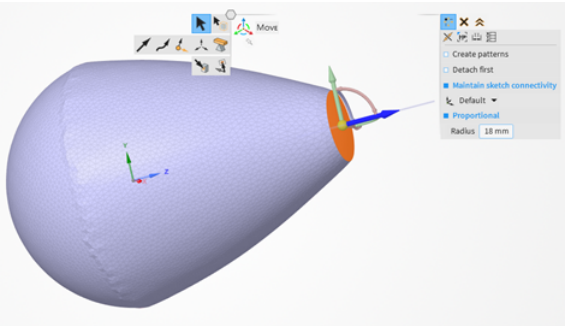
2.6.7. Detect Tool

Two new tools have been added to the **Detect** tool in the **Prepare** tab.

- The **Fasteners** tool detects and classifies parts as bolts, nuts, and washers. The fastener detection method uses a machine-learning–trained heuristic approach.
- The **Sliver faces** tool detects faces within a specified angle range. It identifies sliver faces and supports adding them to a primary selection or converting them into grouped or individual named selections.

2.6.8. Edit Faceted Bodies

When editing faceted bodies using the **Pull** or **Move** tool, the **Proportional** option enables deformation to be propagated proportionally to a region of the faceted mesh, rather than exclusively to the selection.

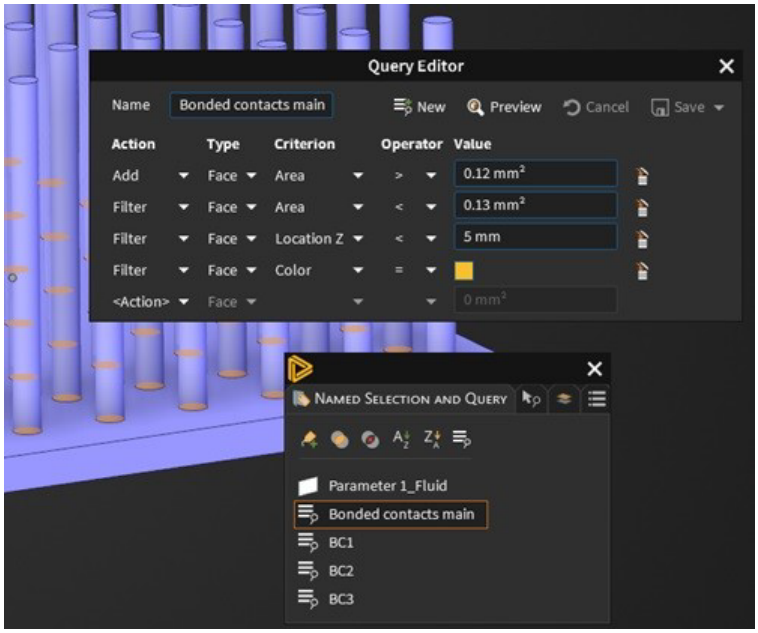


Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

In contrast to the default transformation (extrusion without modifying connectivity or adding new vertices), this option uses the selected regions and the specified angle range which may impact meshing downstream.

2.7. Query Editor

The **Query Editor** enables you to create queries for obtaining a selection set which can be associated with a named selection.



- Enter data in the Query Editor rows to define a selection set, then refine the selection by adding, removing, or filtering entities from the selection.
- Update query-based selections when the scope changes or new bodies are added.
- Transfer the named selections downstream to flagship applications.

Note:

The Query Editor is not available when History tracking is active.

2.8. History Tracking

Enhancements to History Tracking include improvements to the sketch blocks, support for additional tools, and improvements to the geometry parameterization workflow.

- The Sketch block has been streamlined for clarity.
- The Bend and Wrap tools are now supported.
- Geometry parameterization and modification with the Pull-Parameterization and Move-Parameterization tools are now improved. The history blocks recorded for these operations are now distinct.

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

2.9. Materials

Six new records have been added for the 2026 R1 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.

2.10. User Interface Enhancements

2.10.1. Default UI Themes

The Discovery application now includes enhanced versions of the classic **Dark** and **Light** UI themes. These themes prioritize the visual acuity of the model and results, standardize color use, improve contrast, and resolve past inconsistencies. You can return to the legacy themes by navigating to **General > User Interface > UI theme** and choosing **Revert to legacy theme colors**. You can also enable and disable the **Solid background** setting. When enabled, a solid color without any gradient is applied to the graphics scene. Click the color picker to choose a different solid color from the palette.

2.10.2. Beams Assignment

Beams Assignment now supports multiple functions, including **Assign Beam** and new features **Create Vertex Chain** and **Create Vertex Pair**, which simplify the creation of beam geometry and the application of beam profiles. You can assign standard or custom profiles to curves, generate beam paths by chaining vertices, or create connecting curves between vertex pairs.

2.10.3. Extension Builder

The **Extension Builder** now supports shortcut assignments for custom actions. It enables faster access to user-defined tools and commands, improving workflow efficiency.

2.10.4. Export to SimAI

Under **File Menu > Save As**, a new **SimAI** option has been added. This generates a zip containing the .**vtp**, .**vtu** and .**json** files. The .**json** file contains all the numeric meta data for boundary conditions and materials.

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

2.10.5. Import Granta Cloud Materials

When you sign in to the cloud during a session, you can now import Granta cloud materials. Click **Materials > Import from Granta Cloud Materials**. The **Granta Material Picker** launches and enables you to choose materials to add to your local Discovery library, which will then be available for simulations.

2.10.6. Welcome Page

The **Graphics performance mode** setting appears on the Welcome splash screen for easier access. Select from **Auto-detect**, **Standard (best all-around)** and **Enhanced (high frame rates)**.

2.11. User Settings Summary

2.11.1. General

The Discovery application now includes enhanced versions of the classic **Dark** and **Light** UI themes. These themes prioritize the visual acuity of the model and results, standardize color use, improve contrast, and resolve past inconsistencies. You can return to the legacy themes by navigating to **General > User Interface > UI theme** and choosing **Revert to legacy theme colors**. You can also enable and disable the **Solid background** setting. When enabled, a solid color without any gradient is applied to the graphics scene. Click the color picker to choose a different solid color from the palette.

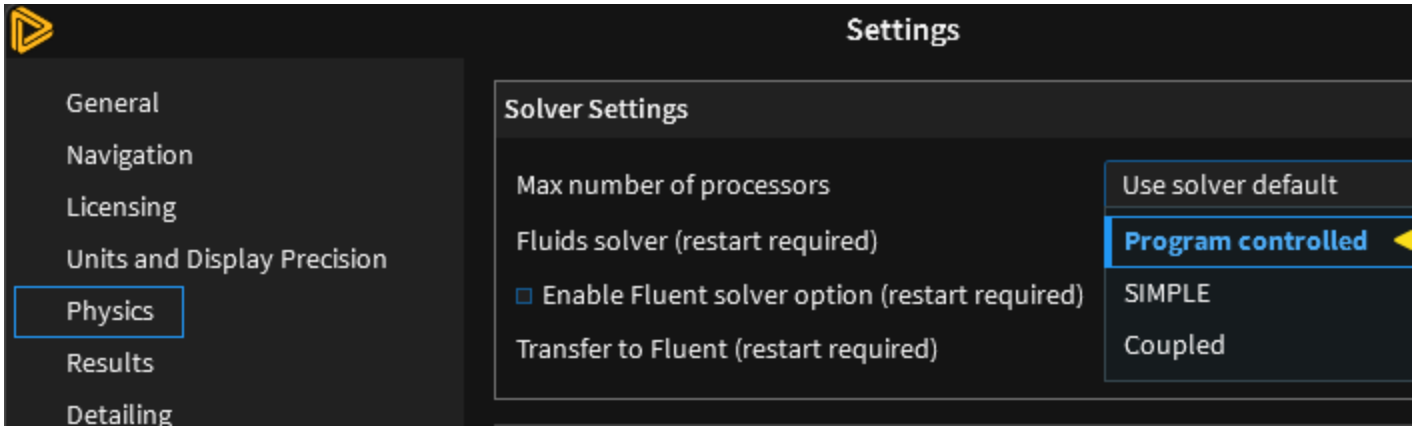
2.11.2. Licensing

You can now change the Discovery application's primary license level without needing to restart the application. To do so, navigate to **Settings > Licensing > Discovery license** and choose the desired level.

2.11.3. Physics

2.11.3.1. Fluids Solver

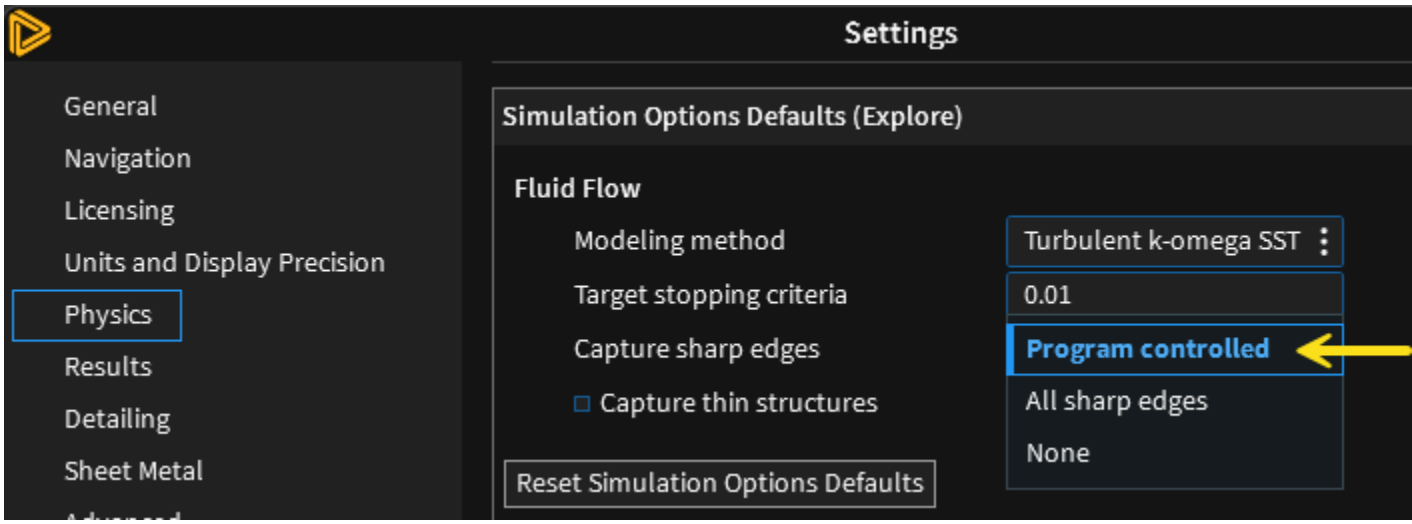
Under **Settings > Physics > Solver Settings**, the default selection is **Program Controlled**, which allows the application to apply the most appropriate pressure-based solver for the simulation. You can specify also **SIMPLE** (a segregated algorithm) or **Coupled** as the defaults.



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

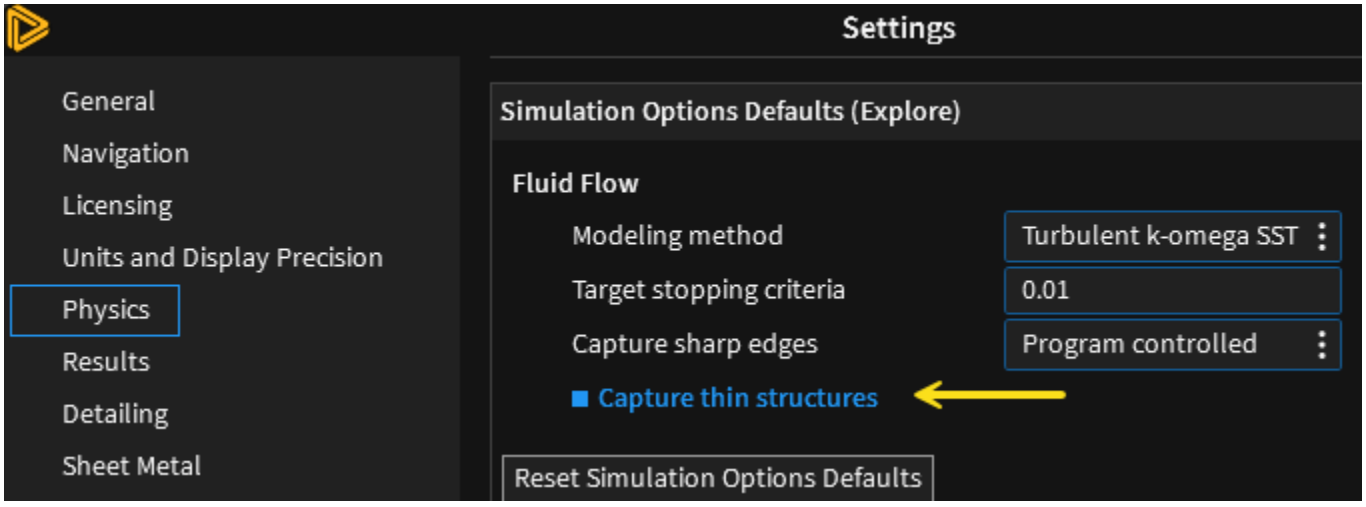
2.11.3.2. Capture Sharp Edges

Under **Settings > Physics > Simulation Options Defaults (Explore)**, the default setting for **Capture sharp edges** is **Program Controlled**. This setting allows the application to choose how to best capture sharp edges on your model. You can change this default setting to instead capture **All sharp edges**, which directs the application to attempt to capture all sharp edges in the mesh. While this option can lead to increased accuracy, it can also increase computation time and decrease solution stability. You can avoid capturing sharp edges by choosing **None**.



2.11.3.3. Capture Thin Structures

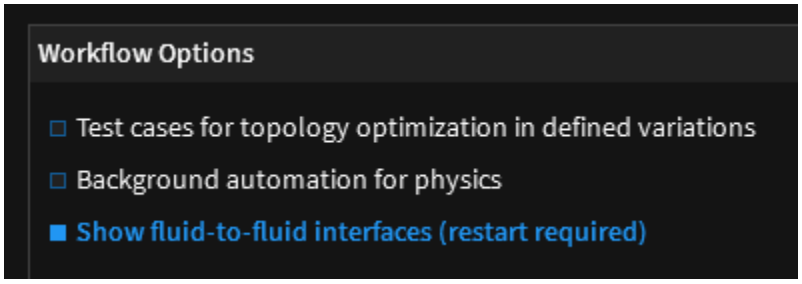
Under **Settings > Physics > Simulation Options Defaults (Explore)**, the **Capture thin structures** option has been added to accurately represent mesh areas with thicknesses smaller than the local size. With this feature, the application creates stretched (anisotropic) cells along thin directions, which can help lower the cell count compared to isotropic refinement. If this option is not enabled, the application generates isotropic cells, and therefore may overlook or merge very thin regions, possibly leading to a loss of detail.



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

2.11.3.4. Fluid-to-Fluid Interfaces

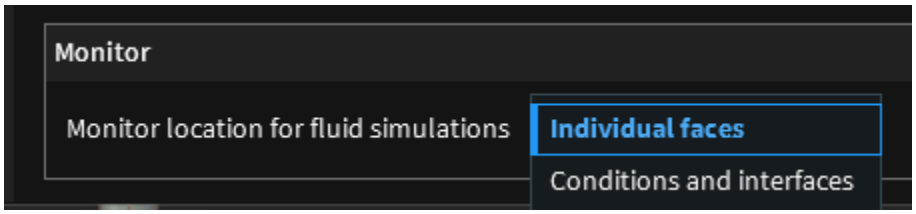
Under **Settings > Physics > Workflow Options**, **Show fluid-to-fluid interfaces** is now selected by default. Fluid interfaces for simulations with fluid-to-fluid interfaces will be visible in the physics tree.



2.11.4. Results

2.11.4.1. Location Setting for Fluid Flow Monitors

Under **Results > Monitor**, you can specify the **Monitor location for fluid simulations** to be on **Individual faces** or on **Conditions and interfaces**. The application calculates results on individual faces only on the selected faces, while calculating results on conditions and interfaces only at the location where the condition is applied.



2.11.4.2. Saved Scenes Remote Download Image Resolution

Under **Results > Saved Scenes**, you can **Set image resolution for remote downloads**. Available resolutions include: Low (1280 x 720), Medium (1920 x 1080), High (2560 x 1440) and 4K (3840 x 2160).

2.11.4.3. Include Explore Results

Under **Results > Full Results**, you can **include explore results** to save **Explore** results to the **.dsco** file. Loading a file that has results will bring the simulation automatically in the **Completed** state, therefore, it is no longer necessary to resolve the simulation to review the results. The results will persist in the file, so the file sizes will be larger.

Select the **Visual results quality** (**High**, **Medium** or **Low**) and **include results variables** if you want to save all or only the essential results.

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

2.11.5. Advanced

When running on a multi-GPU hardware configuration, the new setting **Compute GPU** under **Advanced > Advanced Options** specifies which GPU to use for the solver and may result in improved performance.

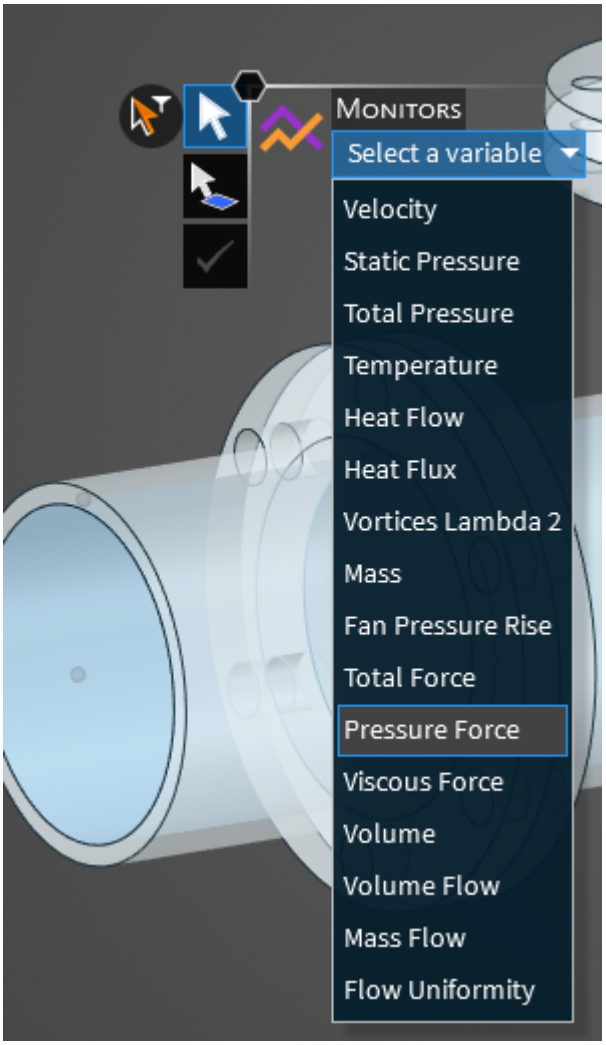
2.11.6. Cloud Services

Under **Cloud Services > General**, you can **Synchronize data with cloud** to maintain GL transmission format binary file (GLB) synchronization between the local Discovery desktop application data and the Discovery web companion application. Options include synching **On save** or not at all if set to **Off**.

2.12. Results

2.12.1. Viscous and Pressure Force Monitors

New force monitors for fluid flow simulations now include **Viscous Force** and **Pressure Force**. Additionally, the **Force** monitor is now named **Total Force**.



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

2.12.2. Wall Yplus Monitors

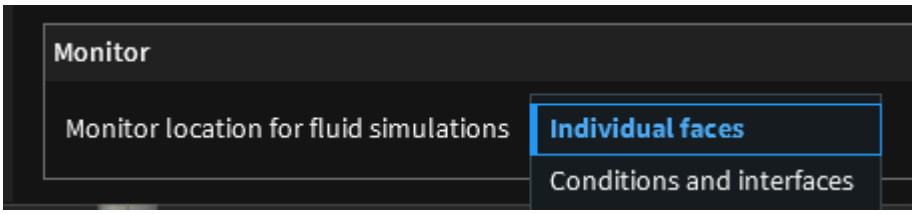
Monitors now include a new wall Yplus monitor result. Yplus indicates whether the near-wall mesh is adequately refined. The value should lie between 30 - 300.

2.12.3. Results Comparison Dashboard

Use the **Results comparison dashboard** when working with saved scenes to capture, download and visually compare results.

2.12.4. Location Setting for Fluid Flow Monitors

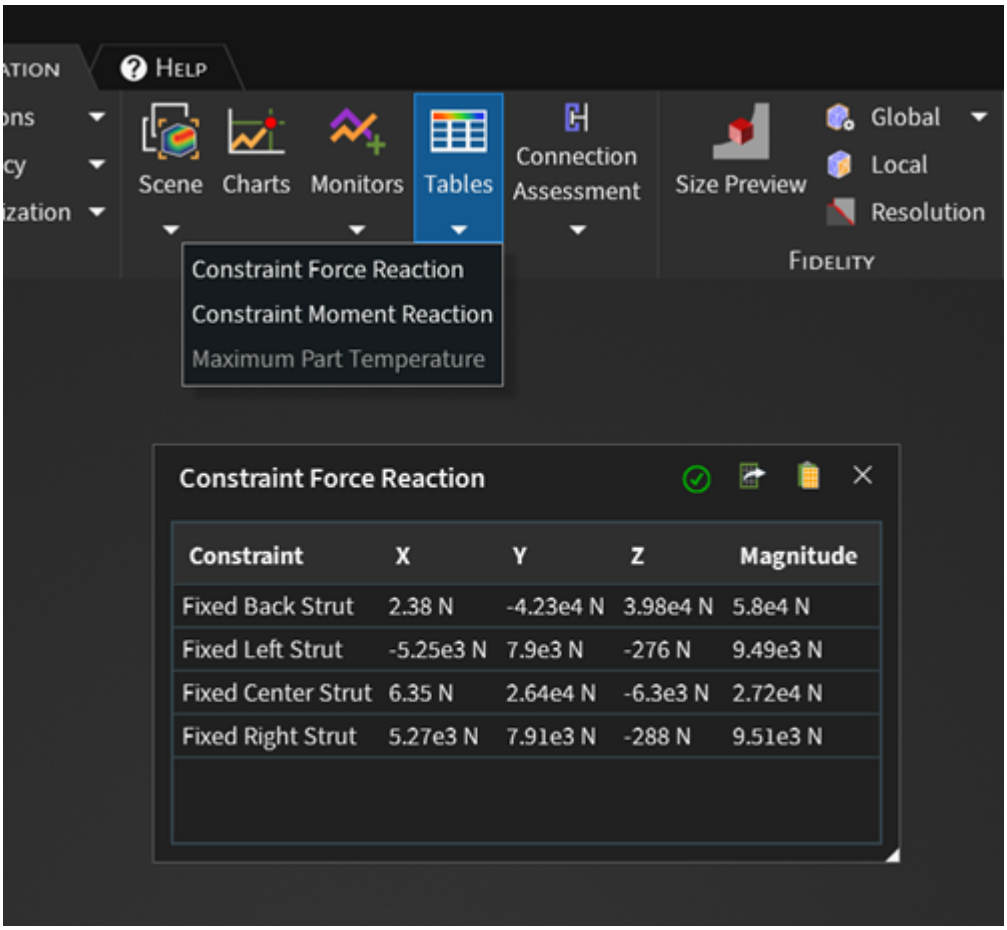
Under **Results > Monitor**, you can specify the **Monitor location for fluid simulations** to be on **Individual faces** or on **Conditions and interfaces**. The application calculates results on individual faces only on the selected faces, while calculating results on conditions and interfaces only at the location where the condition is applied.



2.12.5. Tabular Results

You can now view a tabular summary of Constraint Force Reaction, Constraint Moment Reaction, and Maximum Part Temperature monitors. When viewing your results, you can export the data to a comma-separated or tab-separated list, or you can copy the data to your clipboard as a tab-separated list.

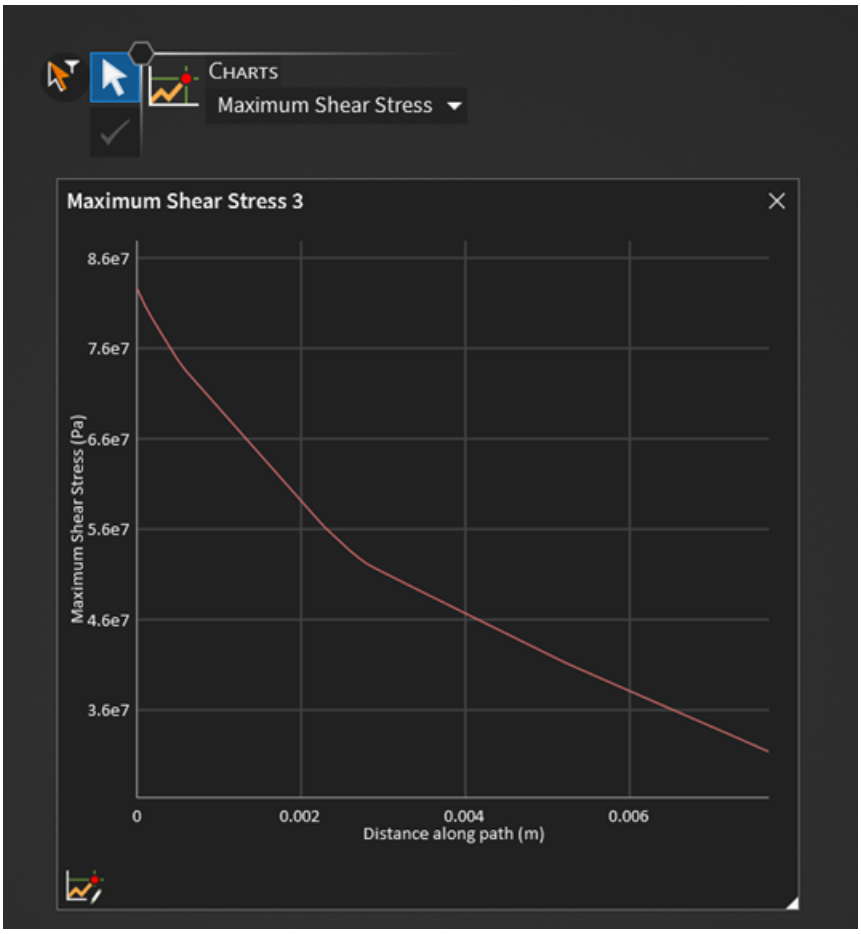
Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.



2.12.6. Structural Chart Results

Structural chart results are now available on curves and edges. You can select a chart location and result variable, then define additional details such as Minimum or Maximum values, and in some cases directional components. You can also reverse the starting point of the path that defines the distance from the X axis to the other end of the curve.

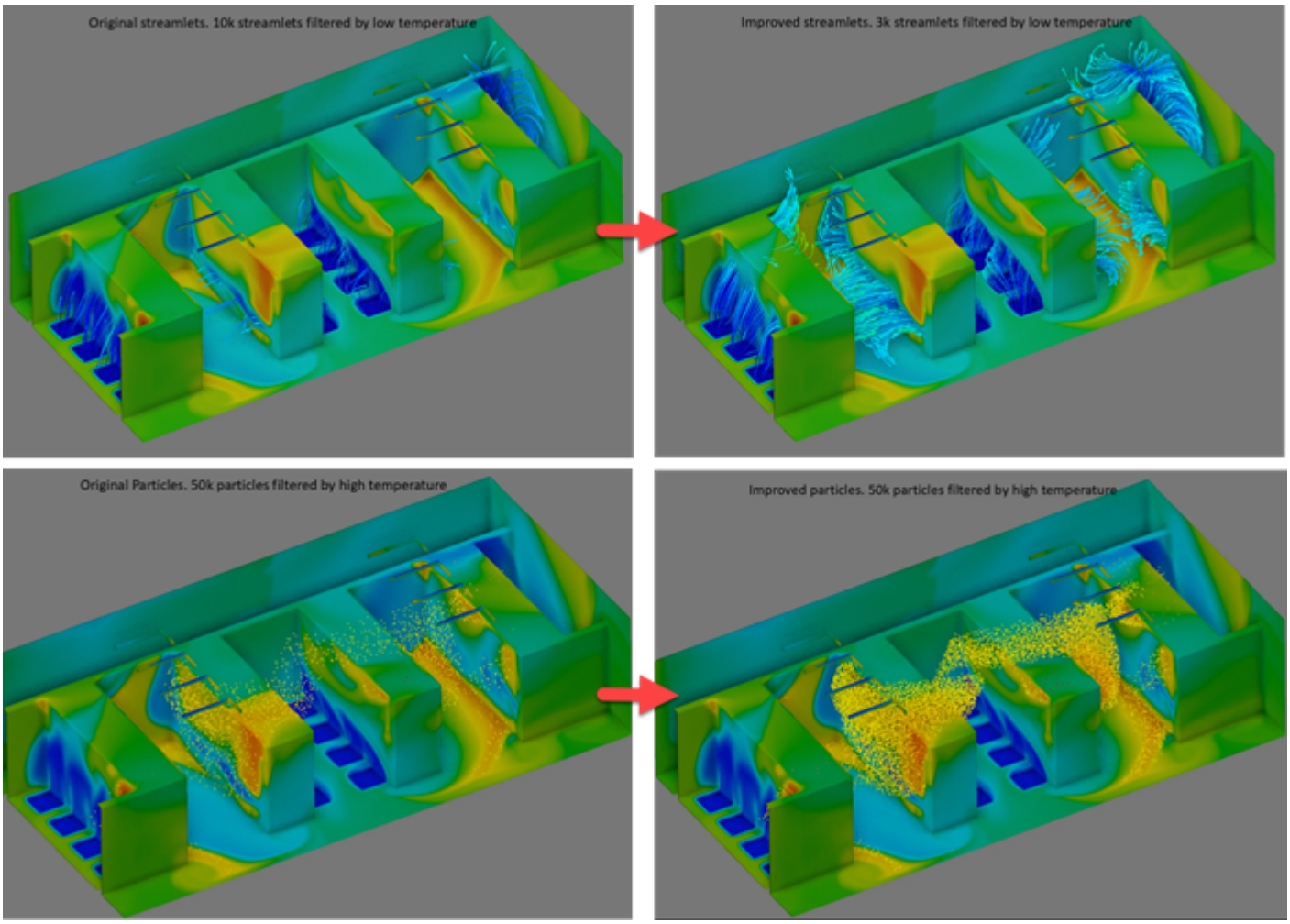
Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.



2.12.7. Filtered Seeding

Improved filtered seeding for streamlets and particles is now available for fluid flow simulations. Previously, reducing the display range would also reduce the number of visible streamlets or particles, making it harder to interpret results. With filtered seeding, particles and streamlets are now seeded directly within the selected range, maintaining consistent counts and improving clarity.

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



2.12.8. Particle Count

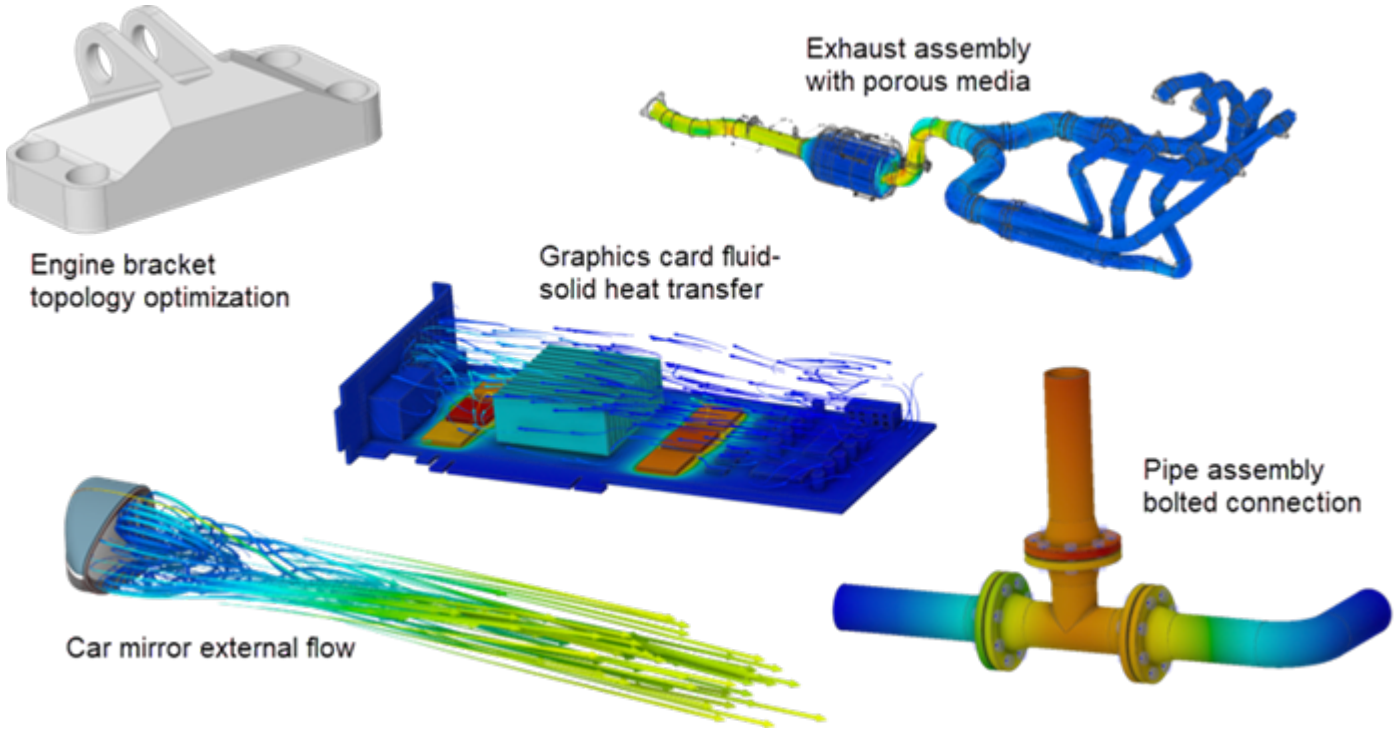
You can now control the particle count for fluid simulations, to aid you in more effectively visualizing the flow.



2.13. Sample Models

The Discovery application now includes new sample models for structures, fluids, and fluid-solid heat transfer.

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



2.14. Performance

This release continues to address performance and responsiveness. You may notice performance improvements in all stages of the application, including:

- Better handling of long-running operations with progress indication, improved messaging and the ability to cancel the operation
- Improved stability using enhanced graphics performance mode
- Increased reliability in opening projects using local fidelity
- Improved GPU control for the LiveGX solver when running on a multi-GPU hardware configuration

2.15. Installation and Licensing

2.15.1. Installer Enhancements

The installer provides the options to:

- Select the **Headless Geometry Service (PyAnsysGeometry)** option to install the Geometry Service packet. For details, see https://geometry.docs.pyansys.com/version/stable/.
- Select from three CAD interface configuration options:
  - **Yes, automatically configure my interfaces**
  - **Yes, I will manually make my selections**
  - **No. Skip configuration. I will configure later**

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

2.15.2. License Preference Overrides

When you first install the Discovery application, and prior to launching it, you can use the DISCOVERY_LICENSE_OVERRIDE system environment variable to override the default ordering of licenses. The application reads this variable the first time it is launched and overwrites the system file that specifies the default license order. Primary and alternate licenses are set accordingly. This enhancement is especially helpful in organizations with mixed license pools, allowing them to dictate which licenses to apply for a specific user.

2.15.3. On Demand License Level Change

You can now change the Discovery application's primary license level without needing to restart the application. To do so, navigate to **Settings > Licensing > Discovery license** and choose the desired level.

2.16. Ansys 3D Design Verification Manual

These verification cases have been added:

- THERM-007 Transient Heat Conduction in a Semi-Infinite Slab
- FLUID-009 Forced Convection Over a Flat Plate
- FLUID-010 Surface to Surface Radiative Heat Transfer Between Two Concentric Cylinders
- FLUID-011 Low Reynolds Number Flow in a Channel with Sudden Asymmetric Expansion
- FLUID-012 Separated Laminar Flow Over a Blunt Plate
- FLUID-013 Turbulent Flow in a 180° Pipe Bend
- FLUID-014 Laminar Flow in a Triangular Cavity

2.17. Ansys Discovery Technology Showcase: Example Problems

This example has been added:

- Mesh Independence Study of a Hip Stem Implant

3. Discovery Application Caveats and Known Issues

At the time of publication, caveats and known issues in Release 2026 R1 of the Discovery application are listed below. Also refer to the *Ansys, Inc. Known Issues and Limitations* document.

- When importing a CREO file created with a version older than CREO 8.0, body names in the model display with an expected ".prt" file type extension. **Workaround**: Resave the CREO file with a newer CREO version. [788192]
- When solving random vibration remotely, 3D interactive results are unavailable. **Workaround**: There is no workaround. [1372995]

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

- The defined electrical conductance is missing when exporting a solid-heat transfer simulation with Joule heating and contacts to the Workbench application. **Workaround**: Specify the electric conductance on the bonded contact in the Workbench application simulation. [1393453]

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

SpaceClaim Release Notes

Release 2026 R1 of SpaceClaim contains the following.

1. SpaceClaim Product Advisories
2. SpaceClaim Changes Overview
3. SpaceClaim Caveats and Known Issues

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. 25).

API Version Support

API version V232 has been deprecated at Release 2026 R1 and will be removed at Release 2027 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 2026 R1:

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:

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

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 v261.
- API version V232 has been deprecated at Release 2026 R1 and will be removed at Release 2027 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 *.scdoc files are imported using a translator. Opening these files may take longer due to the geometry conversion.
- The saving of legacy SpaceClaim *.scdoc files is no longer supported. You can only save files in the *.scdocx format.

3. SpaceClaim Caveats and Known Issues

At the time of publication, caveats and known issues in Release 2026 R1 of SpaceClaim include those listed below. Also refer to the *Ansys, Inc. Known Issues and Limitations* document.

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

- For legacy SpaceClaim (ACIS-based *.scdoc) assemblies, you need to load the *.scdoc based root assembly and save it with 25.x or later versions of SpaceClaim. This ensures that all the referenced component or sub-assembly *.scdoc files are migrated to the newer *.scdocx format during save. The original *.scdoc file is not modified, a copy in *.scdocx format is created. Changes to only component or sub-assembly *.scdoc files do not update the root assembly references to the newer format files. (1349954)

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

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

DesignModeler Release Notes

1. DesignModeler Product Advisories

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

2. DesignModeler Enhancements

DesignModeler 2026 R1 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 2026 R1 of DesignModeler. Also refer to the *Ansys, Inc. Known Issues and Limitations* document.

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

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

Ansys Structural Products

Licensing

At Release 2026 R1, 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. 33)
Mechanical APDL (p. 53)
Autodyn (p. 79)
Aqwa (p. 81)
ACP (p. 85)
Material Designer (p. 89)
Additive Suite (p. 91)
Sherlock (p. 99)
Forming (p. 109)
Motion (p. 119)
LS-DYNA (p. 121)

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. Changes in Product Behavior
2. Documentation
3. Performance
4. Add-ons
5. Graphics
6. Geometry
7. External Model
8. Composites
9. Contact and Connections
10. Tables
11. Mesh
12. Additive Manufacturing Process Simulation
13. Analysis
14. Fracture
15. Substructure Analysis
16. Structural Optimization
17. Motion
18. Imported Loading Conditions
19. Solution
20. Explicit Dynamics
21. Results
22. Scripting

1. Changes in Product Behavior

Release 2026 R1 includes new features and enhancements that result in product behaviors that differ from previous releases. Behavior changes include:

- **Impedance Sheet Formulation for Acoustic Analysis**. The formulation for the **Impedance Sheet** definition has been refined to ensure greater physical consistency in acoustic simulations. See the Mechanical APDL Release 2026 R1 Update Guide for more information.

- **Electromechanical Coupling Coefficient**. When you insert an **Electromechanical Coupling Coefficient** probe in a coupled field simulation that includes **Acoustics** physics, the scoping of the probe now

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

automatically selects the coupled field and structural bodies of the analysis/analyses. Previously, the application automatically used the **All Bodies** option.

- **Imported Body Temperature**. When importing a body temperature load from a result file (`*.rth`) that contains degenerated elements, and 1) the **Mapped Data** property is set to **To Binary File** and 2) the **Weighting** property is set to **Shape Function**, the elements are now classified according to their original element type when the application maps the mesh. The mapping process uses element type classification and shape function weighting algorithms to determine spatial interpolation and node inside/outside criteria.

  Previously, if you had a quadrilateral element with four nodes (1, 2, 3, and 4), and nodes 3 and 4 were duplicates, the application reclassified the element as triangular. Now, the same element is treated as a quadrilateral element, preserving its original type. This change affects interpolation calculations and may result in different mapped target data compared to previous versions. See the Degenerated Shape Elements section of the *Mechanical APDL Element Reference* for more information.

- **Remote Point Application**. Now, when you apply a **Remote Point** to a **single surface node**, the **Behavior** property must be set to **Rigid**. Previously, you could set this property to either **Deformable** or **Rigid**. See the Force-distributed Constraint Theory Background topic in the Surface-Based Constraints section of the *Mechanical APDL Contact Technology Guide* for more information.

- **Remote Points Property Setting When Grouping Loads Imported from Maxwell**. When loads are imported from Maxwell, the **Grouping** property is now set to **To Binary File** by default. As a result, the **Remote Points** property is automatically set to **Globally Available**. The default setting for the **Grouping** property is specified using the **Grouping Default for Imported Remote Loads** (**Options** > Import > **Imported Remote Loads**) preference. Options for this preference include **By Location**, **To Binary** (default), and **No**.

- **Fracture Analysis**. The following fracture-related behavioral changes have been made in the current release:

  – **SMART Crack Growth**: For the **SMART Crack Growth** feature:

    → You can now evaluate the Fracture Probes, **Crack Extension**, and **Total Number of Cycles**, regardless of the setting of the **Store Results At** property (**Analysis Settings** > Output Controls). In previous releases, these results were only supported when **Store Results At** was set to **All Time Points**.

    → The default maximum number of restart files (`*.xnnn`) saved during a restart analysis that includes **SMART Crack Growth** is now **99**, matching the maximum for remeshing database files (`*.rdnn`). Previously, the default maximum was **999** for restart files and **99** for remeshing database files.

  – **Imported Body Temperature**. Previously, during a fracture analysis, when you imported Body Temperature loads (node-based loads) from External Model, and generated a crack mesh over the imported mesh, the application removed scoping if the load was applied to the nodes included in the crack mesh region. Now, the application preserves the scoping and properly maps the load values onto the crack mesh generated on the imported base mesh.

  – **Imported Constraints and Named Selections**. When you import Constraints or Named Selections from External Model into an analysis that includes analytical crack definitions, the application now retains their scoping, as long as these objects are not included in the scoping of

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

crack definitions. In previous releases, such scoping was automatically removed, even if the imported objects were outside the crack mesh zone.

- **Upstream Modal Calculation Improvements for Level-Set and Shape Optimization**. For the Level-Set and Shape Optimization methods of a **Structural Optimization** analysis, the application now performs more accurate result calculations when **Objective** and **Response Constraint** objects are specified for an upstream **Modal** analysis. This enhancement may result in different result values compared to previous releases due to improved computational accuracy.

- **Importing Geometry from Deformation Results**. When you transfer deformed geometry from a supported upstream system to a downstream system in Workbench, the application supports the generated Mechanical APDL common database (`*.cdb`) files, used to transfer the geometry, in COMPACT formats for the **CMBLOCK** and **EBLOCK** commands. See the Coded Database File Commands section of the *Mechanical APDL Programmer's Reference* for more information about these commands. Note that External Model system supports `*.cdb` files generated in COMPACT format.

- **Velocity Result Import from Upstream Coupled Field Harmonic Analyses**. Now, when you import velocities from an upstream **Coupled Field Harmonic** analysis, the **Source Bodies** property of the Imported Velocity object should be set to the bodies that include the velocity results. If the property is set to the **All** option, the application returns an import failure.

  Use the new **All Available Result Nodes** option for the **Source Bodies** property when **Mapped Data** is set to **To Binary File**. This option automatically selects only the bodies in the upstream system that contain result data, streamlining the import process and improving mapping accuracy. By excluding bodies that do not contain result data, it eliminates unnecessary processing operations and improves efficiency.

  Note that you can also select the bodies that include the results using the following options of the **Source Bodies** property:

  – **Manual**: Manually select bodies based on Material ID.

  – **Named Selection**: Select a Named Selections defined in the upstream system.

- **Imported Velocity Mapping**. To improve performance and reduce resource usage when mapping Imported Velocity load data, the application now automatically sets the **Weighting** property to **Triangulation** whenever the **Mapping Control** property is set to **Program Controlled**. Previously, the application used **Shape Function** as the default setting for the **Weighting** property. Unlike **Shape Function**, **Triangulation** uses point cloud-based mapping on nodal data. As a result, it avoids mapping data from interior nodes, which reduces both mapping time and resource consumption.

- **Linked Mode-Superposition (MSUP) Analyses**. Now, when locally (In Process) solving an MSUP Harmonic Response or an MSUP Transient Structural analysis linked to an upstream **Modal** analysis, a new preference, **Enforce Remote File Usage for Modal**, eliminates the need to manually enable the **On Demand Expansion Option** property. When you select the **Enforce Remote File Usage for Modal** preference, the application automatically references prerequisite files using full paths (**RESUME** and **MODDIR**). This reduces setup effort and prevents file duplication, even when **On Demand Expansion Option** property is set to **No**.

- **Connections Group**. When the **Connection Type** property is set to **Contact**, 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 previous releases, the application would create individual contacts. Furthermore, when the **Face/Face** property is set to **Yes** and you change the

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

Group By property to **Faces**, the **Face Overlap Tolerance** is automatically set to **5%**. Previously it was set to **0** (**Off**).

- **Connections Worksheet**. The **Connections** worksheet now includes the following usability improvements:
  - **Table Display and Sizing**: Now, when you first generate worksheet content, the application automatically sizes all table columns to fit to the current screen width.
  - **Default Filter Behavior**: Previously, all property filters were active by default when generating worksheet content. Now, properties with *all* values set to **Program Controlled** are by default, not displayed.
  - **Resetting Columns**: Now, when you select the **Reset Columns** option, the application automatically hides columns with *all* properties values set to **Program Controlled** or Not Applicable (**N/A**).
- **Contact Region Constraint Type**. The **Program Controlled** setting of the **Constraint Type** property now uses **Automatic Constraint Type** for all contacts associated with bodies that have *non-structural* physics. In previous releases, the application used **Distributed, All Directions**.
- **Vector Display** properties for Life and Damage Vector Results in the **DesignLife Add-on** can now be modified from the Mechanical application toolbar.
- For **Strain-Life** analyses using the **DesignLife Add-on**, user-defined Materials Assignment parameters are populated automatically using values from the **Engineering Data**, if available. Otherwise a set of default values is used.
- The **Initial Conditions** object for the Mechanical Motion application has been updated so that it is only applied once, not cumulatively, for geometries within a single part, regardless of how many geometries are selected. If multiple **Initial Conditions** objects are defined for these geometries, only the most recent is applied.
- **Tooltips**: New hover-over tooltips have been added for the Details pane properties of selected objects, including Contact Region, among others
- **Material Optimization Result Display**. In the previous release, results generated using the Material Optimization method were displayed with a transition region between optimized materials, which limited the clarity of the output. This issue has been resolved in the current release. The results now display distinct material regions as intended, improving interpretability and accuracy.
- Python Code objects now support copy, paste, and duplicate.

2. Documentation

The following documentation changes and/or enhancements were made for Release 2026 R1:

- **Contact Documentation**: The contact documentation has been enhanced and reorganized. Definitions and specification-type information are now located in the Contact Region description within the *Mechanical Object Reference*. Engineering context, workflow, guidelines, recommendations, and troubleshooting tips are provided in the Contact section of the *Mechanical User's*

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

Guide. Additionally, new hover-over tooltips have been added to the Contact Region's properties in the Details pane of the Mechanical application.

3. Performance

The following performance related enhancements were implemented for Release 2026 R1:

- **Imported Load Mapping Performance Improvements**.
  - **Processing Time Enhancements**. Now, depending on the number of source and target nodes and the number of time/frequency sets being mapped, the processing time required to import and map load data to your model has been reduced by at least **10%** and as much as **50%**. To accomplish this, the imported load must be specified with the following property settings:

    → **Mapped Data** = **To Binary File**.

    → **Weighting** = **Triangulation** or **Distance-Based Average**, or this property set to **Shape Function** and **Transfer Type** = **Volumetric**.

    Given these settings, the processing improvements occur automatically.

  - **Imported Velocity Mapping**. In addition, to improve performance and reduce resource usage when mapping Imported Velocity load data, the application now automatically sets the **Weighting** property to **Triangulation** whenever the **Mapping Control** property is set to **Program Controlled**. Previously, the application used **Shape Function** as the default setting for the **Weighting** property. Unlike **Shape Function**, **Triangulation** uses point cloud-based mapping on nodal data. As a result, it avoids mapping data from interior nodes, which reduces both mapping time and resource consumption.

4. Add-ons

The following updates were made to the Mechanical add-ons for Release 2026 R1:

- The **DesignLife** add-on offers the following capabilities:
  - A new **Weld Line Creator** worksheet allows you to generate weld definition files for use with solid element models.
  - The **Walker Mean Stress Correction** method is supported for **Stress Life**, **Strain Life**, **Shell Seam Weld**, **Solid Seam Weld** and **Safety Factor** analyses.
  - **Temperature Selection Method** is available for temperature loads in time domain **Stress Life** and **Strain Life** analyses.
  - For **Stress** and **Strain** analyses, the **S-N** and **E-N Curves** defined for the Multi-Curve materials are plotted together with the **Modified S-N Curves** and **Modified E-N Curves** based on the application of surface correction factors.
  - A **Virtual Strain Gauge Graph** post-processing object has been added to validate **Virtual Strain Gauge** time series results.
  - **Damage and Rainflow Histogram Plot** post-processing objects.

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

  - The **Time Series Load Definition** for **Time Domain Loads** includes support for **Sinusoidal** loading.
  - The **View Time Series Plot** option for a **Time Series Load** allows you to display the selected time series file in a graph.

- The **NVH Toolkit** add-on now includes:
  - The **Number Of Paired Modes** calculated for **Result Optimization** in the MAC Calculator.
  - The **Maximum Number of Sensors** in the Sensor Definition for the **Pre-Test Calculator** now also takes into account any manually-added sensors.
  - The **FRF Calculator** offers **Curve-Fitting Analysis** and a corresponding **Stabilization Chart**, displaying all associated information in graphical format.
  - **HDF5** file support for the **MAC Calculator**.
  - Aggressive and Conservative **Filtering Options** for the **MAC Calculator**.
  - Expanded **Visualization Options** for **Mode Animation View**. File 1 and File 2 mode shape results can be superimposed and animated in a single graphical window.
  - An option has been added to the **FRF DOF Pairs Table** to allow computation of the FRF.
  - **Modes Filtering Options** are available in the **MAC Frequency Worksheet**.
- A new **Fast Dynamic Response** add-on has been created, including:
  - The **Fast PSD Response** object, providing a spectrum response for a structural component subject to random excitation.
  - The **Fast Frequency Response** object, providing a frequency response for a structural component subject to a harmonic load.
- Updates for the **Additive Manufacturing** add-ons are described in the Additive Manufacturing Release Notes.
- A new Export To STAR add-on exports the results of an Ansys Mechanical analysis to files that can be loaded into the Ansys Zemax OpticStudio application via the STAR Multiphysics Data Loader. The exported files contain multiphysics datasets, including surface deformation, thermal profiles, and stress files. Export to STAR formats and names these FEA datasets for easy import into the Zemax OpticStudio software. This streamlines the export process and helps you to keep track of the FEA datasets and the surface(s) where they are assigned in an optical design.
- A new Weld Process add-on manages welding process simulations and makes it easier to set up welding models for Static Structural and Transient Thermal analyses. This add-on includes:
  - A **Weld Process** object that lets you configure weld parameters, heat source model, and deposition settings for your welding simulation.
  - A **Weld Setup** object that lets you import and generate a welding sequence, which is displayed in an editable worksheet. You can add multiple Weld Setup objects to simulate parallel welding processes.

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

  - A **Weld Penetration** result object that shows how the welding process affects the temperature profile around the weld regions. You can add multiple Weld Penetration result objects to visualize temperature profiles for different weld regions.

5. Graphics

The following graphical display enhancements were made at Release 2026 R1:

- **New Graphics Preferences**. The application now includes the following new graphics-based preferences (Options > Graphics):

  - **Use Mesh Manager for Mesh Display**: This preference enables internal processing improvements through refactored mesh display logic. While the changes are largely structural, they enhance performance and responsiveness when displaying the model's mesh. Specifically, when you select the **Show Mesh** option, the underlying refactoring enables the application to display and hide bodies significantly faster. Options include **Yes** (default) and **No**.

  - **Apply Body Translucency to Edges**: When enabled, model edges appear transparent, matching the translucency of the associated body/faces. This feature improves visibility of annotations in complex models that contain many parts, making it easier to review and interpret details. Options include **Yes** and **No** (default).

  - **Display All Element Normals in Standard Graphics Mode**. This preference works only when **Display Element Normal for Mesh** is enabled. For objects that do not use Accelerated Graphics, such as the **Connections** object, displaying element normals can take extra time. To improve performance, the preference automatically reduces the number of element normal arrows shown. Enable this new option to display all element normal arrows. Note that for models with a large number of elements, display performance may be slower. Options include **Yes** and **No** (default).

  - **Graphics Display Progress Bar**. The application displays a progress bar while computing the graphics required to display complex meshes and results. Options include **Yes** (default) and **No**.

6. Geometry

The following geometry enhancements were made at Release 2026 R1:

- **Keep Fluid Cross Area on Update/Refresh**. For a line body geometry, when you set the **Model Type** property to **Thermal Fluid**, there is a new preference, **Keep Fluid Cross Area on Update/Refresh** (Options > Geometry), that enables you to prevent the application from resetting the value if you update the source geometry. Options include **Yes** and **No** (default).

- You can now add multiple Body Merge objects to a model. Individual Body Merge objects can be nested within a larger Body Merge, making it easier to assemble complex parts.

  When you add a Body Merge, the Mechanical application creates a Body Merges group in the Model tree (if one is not already present), then adds a Body Merge object within the group. The new Body Merge object is automatically assigned a number (#1 Body Merge, #2 Body Merge, and so forth) in the order that it is created.

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

7. External Model

The following External Model system enhancements were made at Release 2026 R1:

- **External Model Browser**. For the Windows operating system, the new **External Model Browser** window enables you to view the processed and unprocessed keywords/commands for supported file types imported into the Mechanical application using **External Model** options. The browser displays a tree of supported keywords/commands and options that enable you to preview properties or to add a keyword or command to the analysis through a command object.
- **Imported MPC and Joint Connections**. You can now import mesh-based Multi-Point Constraints (MPCs) from Mechanical APDL common database (`*.cdb`) files using the **External Model** system.
- **Imported Shell Thicknesses**. You can now import variable shell thickness data for ABAQUS, NASTRAN, LSDYNA, and Mechanical APDL common database files. The application supports variable thickness on each corner node. Once imported, the **Worksheet** for the **Shell Thicknesses** object provides options that display the thickness values for each node.
- **New Pyramid Element Type Support**. You can now import the following pyramid elements:
  - ABAQUS: C3D5 (linear) and C3D13 (quadratic)
  - NASTRAN: CPYRAM (with the option of being linear)

8. Composites

The following composite related enhancements were made at Release 2026 R1:

- **Composite Sampling Point Tool**. You can now scope the **Composite Sampling Point Tool** to an ACP Solid Model. This enhancement lets you access results from the bottom to the top of the laminate and visualize the through-thickness distribution of key data, including stacking sequence, strains, stresses, and failure indicators. Understanding this distribution is essential for analyzing laminate behavior and optimizing composite designs.
- **Imported Reinforcements**. Mechanical now supports solid transfer with reinforcing plies from Ansys Composite Prep™ (ACP) system. The application imports the reinforcing plies and adds them under the **Imported Reinforcements** group object. A typical application is to model complex layered composite structures such as engine fan blades or 3D printed structures.

9. Contact and Connections

The following contact and connection enhancements were made at Release 2026 R1:

Contact Enhancements

- **Ramping Contact Stiffness**. You can now control how contact stiffness is ramped during your analysis using the new **Ramp Stiffness By** property of the Contact Region object. This property enables you to specify ramping using the following options:
  - **Program Controlled**: The solver decides how and when to ramp the contact stiffness.

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

  - **Load Step**: Stiffness is ramped during a designated load step.
  - **Time Range**: Stiffness is ramped over a specified time interval.

- **New Target Shell Face Option**. Now, using the new option, **Top and Bottom**, of the **Target Shell Face** property, you can model contact pairs on the top and the bottom of a shell/surface body face using only one set of the target surface. This capability reduces the total number of target elements while also improving the processing of large assembly models where the orientations of underlying shell elements are not obvious or consistent.
- **Constraint Type Enhancements**. The **Constraint Type** property option, **Mixed Assembly**, now supports solid-to-solid contact definitions.

Connections Enhancements

- **Joint, Spring, and Beam Connections**. When defined as a **Remote Attachment**, you can now scope Joints, Springs, and Beam connections to one or more nodes.
- **Automatically Merge Contacts**. When using the Connections Group object with the **Connection Type** property set to **Contact** and the **Group By** property set to **Faces**, the application now automatically merges contacts wherever possible. This enhancement enables you to:
  - Reduce over-constraint issues found with MPC formulation-based contacts, particularly when multiple contacts share the same target faces or have overlapping scoping.
  - Minimize the number of contacts created in the majority of the cases. For example, a PCB model where a multiple contacts share a common target face. This also reduces the overall computational time taken by the solver in most cases.
- **Connections Worksheet**. The **Connections** worksheet now includes a new context (right-click) menu option: **Fit to Screen Width**. This feature automatically adjusts the widths of all table columns to evenly fill the available space.

  Additionally, when you select the existing **Reset Columns** option, the application now automatically hides any column where all values are set to **Program Controlled** or Not Applicable (**N/A**).

10. Tables

For Release 2026 R1, the following table enhancements have been made:

- New independent and dependent variables are available in the Tables function to support Mechanical APDL scripts that contain the ***DIM** command. These new variables include field variables and dimensionless numeric data.

11. Mesh

For mesh related enhancements, see the Meshing release notes.

12. Additive Manufacturing Process Simulation

See Additive Manufacturing.

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

13. Analysis

Release 2026 R1 includes the following analysis enhancements:

- **Mode Superposition (MSUP) Transient Structural Analysis**. You can now use the Load Application feature to apply and scale base excitations or load vectors, generated by an upstream Modal analysis, in a downstream MSUP **Transient Structural** analysis.
- **Load Support for Modal Analyses**: You can now specify the following loading conditions in a **Modal** analysis:
  - Standard Earth Gravity
  - Pipe Pressure
  - Bearing Load
  - Line Pressure
  - Imported Body Force Density
  - Imported CFD Pressure
  - Imported Pressure

  These loads do not affect natural frequencies or mode shapes. However, you can use the load vector data from these boundary conditions. Apply and scale them with the Load Application object for downstream Mode Superposition (MSUP) Harmonic Response and/or MSUP Transient Structural analyses.
- **Apply Commands to Load Vectors Using a Commands (APDL) Object**. During Modal or Substructure Generation analyses, you can now use the Commands (APDL) object to apply command snippets to specific load vectors using its new **Load Vector Selection Mode** property. The options of this property enable you to apply the **Commands (APDL)** object to all load vectors or a specific load vector of the analysis. Options include **All** (default) and **By Number**. This feature requires you to insert the command object under the **Environment** object of the given analysis.
- **Multistage Cyclic Symmetry**. For stand-alone **Static Structural** analyses, the application's multistage cyclic symmetry capability now supports the following boundary condition and loading definitions:
  - You can now apply Remote Force, Moment, and Remote Displacement boundary conditions within multistage cyclic symmetry setups.
  - A Rotational Velocity can be specified using a harmonic index value, as defined by the Stage object.
- **Analysis Settings** > **Step Controls**. A new property, **Maximum Iterations Per Substep**, has been added to the Step Controls category of the Analysis Settings object. For nonlinear static and transient analyses, this property enables you to specify the maximum number of iterations the solver can perform within each substep of a nonlinear analysis to reach equilibrium.
- **Analysis Settings** > **Rotordynamics Controls**. The **Coriolis Effect** property now supports the **On, Rotating Reference Frame** setting for **Harmonic Response**, **Modal**, and **Static Structural** analyses.

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

- **Coupled Field Analyses**. When coupling structural and thermoelectric conduction analyses, the application now uses the SOLID225 element for lower order meshes. Previously, it used SOLID226, a higher-order element, but dropped the mid-side nodes. This change ensures more appropriate element selection for the analysis type.

14. Fracture

Release 2026 R1 includes the following Fracture analysis enhancements:

- For the **SMART Crack Growth** feature:
  - **Fracture Probes**: You can now evaluate **Fracture Probes**, **Crack Extension** and the **Total Number of Cycles**, regardless of the setting of the **Store Results At** property (**Analysis Settings** > Output Controls). In previous releases, these results were only supported when **Store Results At** was set to **All Time Points**.
  - **Restart Controls**: A new property, **Maximum Total Files to Save**, has been added to the **Restart Controls** category of the Analysis Settings object. This property defines the maximum number of restart files that can be saved during a restart analysis. Use this property to control the total number of restart files retained for future restarts of the analysis. During a **SMART Crack Growth** restart analysis, the default maximum number of saved files is **99** for both restart files (`*.xnnn`) and remeshing database files (`*.rdnn`).
- **Imported Body Temperature**. Previously, when you imported body temperature loads (node-based loads) from External Model, and generated a crack mesh over the imported mesh, the application removed scoping if the load was applied to the nodes included in the crack mesh region. Now, the application updates the scoping and properly maps the load values onto the crack mesh generated on the imported base mesh.
- **Imported Boundary Conditions and Named Selections**. When you generate a crack mesh on an imported base mesh that includes imported Contacts, Constraints, Surface Loads, and Named Selections from an external mesh model, the application now retains their scoping, as long as the scoping of these objects is unaffected by the crack mesh generated locally. In previous releases, such scoping was automatically removed, even if the imported objects were outside the crack mesh zone.

15. Substructure Analysis

Release 2026 R1 includes the following Substructure Analysis enhancements:

- **Generation Pass Enhancements**. When defining a Condensed Part or performing a Substructure Generation analysis:
  - You can now control whether the Constraint Mode Data (CST) required for an Expansion Pass is written to the Component Mode Synthesis (`*.cms`) file using the new **Future Expansion** property. Omitting the Expansion pass related data significantly reduces the size of the `.cms` file which is ideal for workflows where the expansion pass is not needed.
  - The application does not generate the `*.LN22` and the `*.BCLV` files during the Generation pass except when the **Keep Files For** property of the **Condensed Part** object is set to **MAPDL Expansion**.

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

16. Structural Optimization

Release 2026 R1 includes the following Structural Optimization analysis enhancements:

- **Cyclic Symmetry**. The optimization process now includes Cyclic and Pre-Meshed Cyclic symmetry types during shape derivative computations for the following optimization methods:

  – Topology Optimization - Mixable Density

  – Topology Optimization - Level Set Based

  – Shape Optimization

  This enhancement allows you to take advantage of the computational efficiency offered by symmetry-based analysis, reducing model size and solution time while maintaining accuracy.

- **Millability Manufacturing Constraint**. A new **Manufacturing Constraint** is available: **Millability**. Use this manufacturing constraint to design optimized structures that you can manufacture using four-axis milling.

- **User Defined Criteria for Materials Optimization**. For the **Materials Optimization** method, you can now specify user-defined criteria:

  – On geometry using the Measures object.

  – In an upstream **Static Structural** analysis using the Primary Criterion and Composite Criterion objects.

  You can then specify this criterion in the Objective and Response Constraint objects.

- **Member Size Manufacturing Constraint**. The application now supports the **Maximum Size** and **Gap Size** properties of the **Member Size** manufacturing constraint when using the **Topology Optimization – Mixable Density** method.

- **Materials Optimization Results**. For the Materials Optimization method, the result type depends on the selected formulation. For this method, when you use the specify multiple materials as **Full** (**Filling Method**), the application automatically creates a Topology Multi-Density result. When you specify **Lattice**, the application automatically creates a Lattice Density result.

17. Motion

The following enhancements were made to the Motion application at Release 2026 R1:

- The Motion application now offers **System Coupling** with the **Maxwell** application, using a **Motor EM Force** object that supports both 2D and 3D designs. You can also specify the **Maxwell** application version to ensure solution consistency in **Magnetic Field** analysis.

- **Co-simulation** between the Motion and **Maxwell** applications is also now supported, using the **Magnetic** object to define the groups of bodies that require magnetostatic analysis.

- **Output Controls** have been updated to allow transfer of coordinate systems to the Motion standalone preprocessor, and enable or disable recording of **EHD Bearing** and **Motor EM Force** results.

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

- The **Joint Load Properties** object now supports **Point on Curve** (PTCV) joints. PTCV joints also now support velocity and acceleration for sliding motion.

- A new **Export Motion Files** dialog has been added to allow selective export of DFS, SSC and DFSXML files, as well as reference and post graphic files.

- The **Drivetrain Toolkit** includes a **Contact Damping Ratio** property for **Bearing** objects, **Gear Configuration** for **Planetary Gearsets** without sun and ring gears, a range of **Tolerance** options for **External Gearsets**, a **Tooth Pressure** property for the **Drivetrain Assembler**, and a **Gear Summary** of the latest **Gear Set** specification including an **Export** button to save this as an HTML file.

- The **Links Toolkit** now supports the use of **Multibody Parts**, includes a **Synchronize Stiffness** option for the **Links Assembler** to avoid errors due to the modification of the stiffness behavior during analysis, and an **Engage** option to provide manual control of the wrapping method for the **Path** object. Dummy contacts that are generated internally before cloning and are now no longer displayed in the **Links Assembler** tree.

For changes to the Motion solver, see the Motion Standalone Release Notes.

18. Imported Loading Conditions

The following imported loading condition enhancements were made at Release 2026 R1:

- **Imported Load Mapping Improvements**.

  – **Processing Time Enhancements**. Now, depending on the number of source and target nodes and the number of time/frequency sets being mapped, the processing time required to import and map load data to your model has been reduced by at least **10%** and as much as **50%**. To accomplish this, the imported load must be specified with the following property settings:

    → **Mapped Data** = **To Binary File**.

    → **Weighting** = **Triangulation** or **Distance-Based Average**, or this property set to **Shape Function** and **Transfer Type** = **Volumetric**.

  Given these settings, the processing improvements occur automatically.

  – **Imported Velocity Mapping**. In addition, to improve performance and reduce resource usage when mapping Imported Velocity load data, the application now automatically sets the **Weighting** property to **Triangulation** whenever the **Mapping Control** property is set to **Program Controlled**. Previously, the application used **Shape Function** as the default setting for the **Weighting** property. Unlike **Shape Function**, **Triangulation** uses point cloud-based mapping on nodal data. As a result, it avoids mapping data from interior nodes, which reduces both mapping time and resource consumption.

  – **Progress During Load Import and Mapping**. Also, when you are using the **To Binary File** option of the **Mapped Data** property to import a load, the application now displays a progress bar that indicates the current frequency or time set being interpolated, along with the total number of sets to process.

  – **Parallel Processing for Binary File-Based Imported Loads**. To improve the data mapping process on Linux, you can now set the maximum number of cores using the Workbench

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

Maximum Number of Cores preference (**Tools** > **Options** > Mechanical). By default, the application uses all cores. This capability requires the following settings:

→ **Mapped Data** = **To Binary File**

→ **Weighting** = **Shape Function**

→ **Transfer Type** = **Volumetric**

- **Specifying Source Bodies for Imported Loads**. For Imported Velocity and Imported Body Temperature loads, when **Mapped Data** = **To Binary File**, the **Source Bodies** property now includes a new drop-down option: **All Available Result Nodes**. This option automatically identifies and selects all bodies in the upstream system that contains result data. This enhancement eliminates the need to manually define Named Selections when the upstream result is filtered, simplifying setup and improving data mapping efficiency.

- **Imported Heat Generation from Maxwell**. You can now import Imported Heat Generation loading conditions from the Maxwell application to a downstream Coupled Field Static or Coupled Field Transient analysis.

- **Imported Body Temperature**. When importing data for an **Imported Body Temperature** load with **Mapped Data** set to **To Binary File**, the application now updates only the modified **Scale** or **Offset** values (as adjusted in the **Data View** window). It no longer remaps all load data. This targeted update significantly reduces mapping time for **Scale** or **Offset** changes, especially when working with moderate to large source datasets.

- **Importing Body Temperature Using External Data**. For Static Structural and Transient Structural analyses, you can use External Data to import temperature results in **H5DPF** file format and define an Imported Body Temperature load. This file type is created using ANSYS Data Processing Framework (DPF) and is based on the Hierarchical Data Format version 5 (`*.h5`).

- **Grouping Loads Imported from Maxwell**. Now, when importing remote loads from the Maxwell application, use the **Grouping Default for Imported Remote Loads** (**Options** > Import > **Imported Remote Loads**) *preference* to change the default setting for how the application groups the load data using the **Grouping** property. Options for this preference include **By Location**, **To Binary** (default), and **No**.

19. Solution

Release 2026 R1 includes the following solution enhancements:

- **Ansys HPC Platform Services (HPS) Improvements for Structural Optimization**

  Mechanical now supports direct referencing of result and database files from upstream analyses when performing Structural Optimization with HPS Solve.

  – When solving with HPS, Mechanical no longer downloads and reuploads result (`*.rst`) and database (`*.db`) files from upstream analyses to downstream Structural Optimization systems.

  – These files are referenced directly on the server, reducing costly file I/O operations.

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

  - After the **Structural Optimization** solution completes, the `*.rst` file no longer must be downloaded for post-processing. Mechanical accesses it directly.

  Display File Download Rate While Downloading Results

  In addition, when downloading files from HPS, Mechanical now displays:

  - The current file download rate.
  - Individual file sizes.
  - The total amount downloaded during the operation.

  This visibility helps you better understand transfer performance and progress when working with large result files.

- **Resource Prediction**. This feature:
  - Now incorporates hardware performance characteristics from the HPC Platform Services (HPS) queue, when available, to provide more accurate estimates for solution times and memory usage.
  - Has been enhanced to display a prompt recommending the use of the GPU acceleration capability in order to improve performance.
- **New Mixed Solver Type for Structural Analyses**. A new solver type, **Mixed**, is now available for Static Structural and Transient Structural analyses. This option combines the strengths of both the sparse and PCG solvers, offering improved speed and memory efficiency without compromising accuracy. See the **EQSLV** command in the *Mechanical APDL Command Reference* for more information.
- **Solution Statistics**. When you choose to use GPU acceleration, the **Solution Statistics** worksheet (Solution Information > **Solution Output**) now provides new GPU related information in the **HPC Metrics** section of the worksheet, such as the selected card and whether GPU acceleration was beneficial for the solution.

20. Explicit Dynamics

The Workbench application provides multiple systems that enable explicit solutions:

- Explicit Dynamics
- LS-DYNA

The enhancements for these systems for Release 2026 R1 are described below.

Explicit Dynamics

The following updates were made at Release 2026 R1:

- **Ansys HPC Platform Services (HPS)**. HPS can now be used with the Explicit Dynamics system as an alternative to the Remote Simulation Manager (RSM) for creating and managing jobs.

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

LS-DYNA

For LS-DYNA systems, note the following enhancements for Release 2026 R1:

- **1D ALE Analysis Support**. You can now run a 1D ALE analysis for a system that has 2D behavior. Line body properties can be set using the Section object, which then writes the *SECTION_ALE1D card to the input file.
- **2D Postprocessing by Material**. You can now plot results based on an S-ALE material for a 2D analysis in the LS-DYNA system.
- **ALE Mesh Motion Object**. The new ALE Mesh Motion object can be used to apply motion to an underlying mesh of bodies that have an S-ALE Domain **Reference Frame**.
- **Bushing Joint and General Joint Support**. The LS-DYNA system now supports the use of Bushing Jointsand General Joints for modeling.
- **Custom Time Step Size Object**. The new Custom Time Step Size object allows you to define a time-step curve using tabular data, specifying how the time-step size changes over time. This data can then be applied to step controls in the **Analysis Settings**.
- **Cylindrical Coordinate System for Displacement**. You can now use a cylindrical coordinate system on a set of nodes to restrain the degrees of freedom (radial, tangential and axial).
- **D3dlrf Support**. To assess if dynamic relaxation was successful, the LS-DYNA system now supports *DATABASE_BINARY_D3DRLF to postprocess the dynamic relaxation result file.
- **Deformable to Rigid Switch Type**. The **Switch Type** in a Deformable to Rigid object, defines how the deformable to rigid switch is performed. It can be set to Manual, where the transformation takes place at the start of a simulation, or it can be Automatic, where the switch is triggered at a specified time during the calculation.
- **Editable Multiple RPMs Analysis Properties**. A Multiple RPMs analysis can be changed from having read-only RPM properties to editable RPM properties.
- **Eroding Contact Program Controlled Values**. When the contact **Type** is set to Eroding for a Contact Properties object, the Eroding Controls (**Symmetry Plane Option**, **Erosion Interior Node Option** and **Solid Elements Treatment**) are now set to 1 in the solver when Program Controlled is selected. This means faces with normal boundary constraints (such as segments of brick elements on a symmetry plane) are not included, storage is allocated so that eroding contact can occur, and solid element faces are included if they are on the boundary of the material subset.
- **Full Support of Python Commands**. The Get Solve Commands option is now available as a **Target Callback** in the Python Code object for the LS-DYNA system, providing the same support as in Mechanical.
- **Importing Rigid Bodies**. LS-DYNA rigid bodies can be imported through External Model. Once imported, they are defined as rigid bodies with full meshes in Mechanical.
- **Importing Contacts**. LS-DYNA contacts that are imported through External Model are now scoped to bodies instead of element faces.

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

- **Internal Heat Generation Support for Thermal Workflows**. The LS-DYNA system supports the Internal Heat Generation thermal loading condition and can be applied using the *LOAD_HEAT_GENERATION keyword.
- **Kelvin Support**. If a Metric system is selected for the **Solver Unit System** under the Analysis Settings, you can set the **Solver Temperature Units (For Metric Systems)** to either Celsius (C) or Kelvin (K).
- **Keyword Snippet Improvements**. When typing keywords in a Keyword Snippet (LS-DYNA) object, there is now a drop-down list suggests supported keywords based on your input. After the keyword has been entered, the Commands worksheet automatically populates with the keyword's parameters and default values. Additionally, you can now use the *DEFINE_CURVE keyword in a Keyword Snippet (LS-DYNA) object.
- **Linux Support**. The LS-DYNA External Model can now be used on Linux operating systems.
- **LS-DYNA Version**. 16.1 is the LS-DYNA solver version included in Release 2026 R1.
- **Material Support**. The material models *MAT_TABULATED_JOHNSON_COOK and *MAT_SPOTWELD are now supported in the LS-DYNA system.
- **Multibody Parts Support**. You can now create multibody parts that consist of both rigid and flexible parts.
- **Pressure Result for ALE Tracer**. You can now view the pressure (**p**) result generated by an ALE Tracer object.
- **Python Command Support**. Get Solve Commands is now available as a **Target Callback** option in the Python Code object, meaning the LS-DYNA system now provides the same options as Mechanical.
- **RAND_I in User Defined Results**. The electric current of a Randles circuit (RAND_I), is now available as a user defined result.
- **Radiation Support for Thermal Workflows**. The LS-DYNA system supports the Radiation thermal loading condition and can be applied using the *BOUNDARY_RADIATION keyword.
- **Remote Point Scoping** In the Velocity boundary condition, you can now scope to any pre-defined remote points that have a Rigid **Behavior**.
- **Results Scaling**. In the Result ribbon, the **Deformation Scale Factor** is now set to 1.0 (True Scale) by default. This means the deformed shape displayed during post-processing is shown at its true physical scale, which corresponds exactly to the calculation without any exaggeration or reduction.
- **S-ALE Volume Filling for Shell Bodies**. Shell bodies can be filled with a material by setting the Reference Frame to S-ALE Fill and assigning a material.
- **Trim Operations for Mesh Workflow**. S-ALE Mesh Workflows now provide Trim options to shape the ALE mesh to be used in the simulation so that it closely mirrors the geometry.

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

21. Results

Release 2026 R1 includes the following result enhancements:

- **Result Data Caching**. For Modal and MSUP Harmonic Response analyses, when the **On Demand Expansion Option** property (**Analysis Settings** > Options) is set to **Yes**, all result types now include a new property: **Save Auxiliary Results**. When enabled, this property automatically caches result data in memory. This caching significantly improves result retrieval and display performance when switching between objects. The performance improvement is maintained even after you close and reopen a solved analysis.

- **Line Chart**. For LS-DYNA analyses, new features and capabilities are available for the **Line Chart** feature, including:

  – **Charting X Axis Source Data**. The new **X Axis** property enables you to choose a dataset to plot on the x-axis and the remaining datasets are reconstructed to map to this newly specified axis.

  – **Updating the Chart Source Data**. A new **Update** option is now available for when you modify source objects. When selected, the application deletes the datasets of the source object(s) and re-imports the revised data, ensuring the chart reflects the most current information.

  – **New Source Data Types**: Line charts now support the loading conditions Acceleration, Force, and Pressure as source data objects.

  – **Editing Labels**. You can now directly edit the content of a chart label.

  – A new **Chart** tab is now available in the Chart Options window. This tab enables you to customize the chart's display using the following options:

    → **Title**: Enter a title to display at the top of the chart.

    → **Font Size**: Select the font size for the chart title.

    → **Legend**: Toggle the legend on or off.

    → **Position**: Define the position of the legend. By default, this property is set to **Program Controlled**, which positions the legend in the most ideal location based on the data presented in the chart.

  Many of these new features offer associated APIs. See the Working with Line Charts section of the *Mechanical Scripting Guide* for more information.

- **Calculating Midside Node Results**. For the supported solid element types, use the new Options dialog preference, **Compute Midside Node Values Based on Averaged Corners** (**Options** > Results > **Default**), to specify how the application calculates midside node results. For averaged results, set the preference to:

  – **Yes** (default) to first calculate the averaged corner node values and then the midside node result using the calculated averaged corner node values.

  – **No** to calculate midside node values using *unaveraged* corner node values.

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

- **Moment Reaction Probe**. There is a new preference, **Moment Probe Summation Point Offset By Displacements** (Options > Results), which affects the calculation of the moment arm of a **Moment Reaction** probe. With this preference enabled, the application moves the **Summation** point for the probe to match the closest deformed node. Options include **Yes** and **No** (default).

- **Far-field Results on Adapted Meshes**. You can now compute Far-field results in Harmonic Acoustics analyses when using an adapted mesh generated through the Mesh Morphing feature.

- **Displaying Only the Body with Maximum Result Value**. A new context menu option, **Show Body With Max Result**, is now available for all evaluated result objects. When selected, the application automatically displays only the body that includes the maximum value, as determined by the location of the **Maximum Occurs On** property value, for the specified result.

22. Scripting

Release 2026 R1 includes the following enhancements to scripting in the Mechanical application:

- **CPython Support**. The Mechanical scripting function now supports CPython (Python 3.x) in addition to IronPython (Python 2.x). You can select either CPython or IronPython as the Python engine in the Mechanical Scripting Window, the Button Editor, Python Code objects, and Python Results objects. In addition, you can specify the Python engine for Mechanical ACT extensions and for running a script when you launch the Mechanical application from the command line.

  You can set the default Python engine via **File** > **Options** > **UI Options** > **Scripting**. This applies to new scripts, buttons, and Python objects, plus scripts that you run from external files via **Run Macro**. For backwards compatibility with existing scripts, the Python engine defaults to IronPython.

  For more information about using CPython in Mechanical scripts, see the Scripting in Mechanical Guide. For a discussion of the differences between IronPython and CPython, see Migrating from IronPython to CPython.

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

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

Mechanical APDL Release Notes

Release 2026 R1 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. 53)
- Multiphysics (p. 60)
- Solvers (p. 63)
- High-Performance Computing (p. 64)
- Other Enhancements (p. 65)
- Commands (p. 66)
- Elements (p. 68)
- Documentation (p. 70)
- Technology Showcase: Example Problems (p. 70)

For more information about this release, see the Update Guide (p. 71) section of this document and the Global Release Notes.

1. Structural

Release 2026 R1 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

1.1. Contact

Release 2026 R1 includes the following new features and enhancements for analyses involving contact.

1.1.1. Enhancement of GCGEN and GCDEF commands for better performance and reliability
1.1.2. Moment Convergence Check for Pilot nodes for Structures with No Other Rotational DOFs
1.1.3. Automatic Merging of Overlapping Multi-point Constraint (MPC) Contact Pairs
1.1.4. New Command Stabilizes Nonlinear Solutions
1.1.5. CNTGEN and SFGEN Commands Now Support Nodal Components

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

1.1.6. Performance Improvements for Multiple Spot Welds
1.1.7. 2D-3D Bonded MPC Contact for Solid Assemblies Supports Thermal and Structure-thermal Coupling
1.1.8. DMP Performance Improvement of MPC CONTA175 Element
1.1.9. Improved Convergence when Simulating Assemblies that Include Gasket Elements

1.1.1. Enhancement of GCGEN and GCDEF commands for better performance and reliability

The general contact pre-process commands, **GCGEN** and **GCDEF**, have been enhanced to offer improved accuracy and reduced operation speed. Improvements target the main functions of these commands: generating the general contact surface, assigning contact properties, and listing or detailing the properties. These enhancements lead to faster pre-processing times and more reliable contact definitions, particularly useful when working with large assembly models.

1.1.2. Moment Convergence Check for Pilot nodes for Structures with No Other Rotational DOFs

Moment convergence is now checked at the pilot node of TARGE170 when target elements have rotational degrees of freedom (DOFs). Previously, this check at the pilot nodes was ignored if there were no structural elements with rotational DOFs in the model. This improvement improves accuracy.

1.1.3. Automatic Merging of Overlapping Multi-point Constraint (MPC) Contact Pairs

The new **CNCHECK**, MERGE,,,,MPC pre-processing command addresses the issue of overconstraint in MPC bonded contact by automatically merging all possible overlapping contact pairs. Key benefits are improved convergence behavior and a reduction in solution time and memory usage. This new functionality is especially useful for large, complex models with many interconnected parts using MPC bonded contact definitions.

1.1.4. New Command Stabilizes Nonlinear Solutions

If a nonlinear analysis fails in the very first iteration due to pivoting errors (usually for under-constrained problems), use the new **WKSP** command to improve stability and convergence. The **WKSP** command automatically adds weak springs (COMBIN14 elements) at the nodes and DOFs exhibiting near-zero pivots and reattempts the solution.

1.1.5. CNTGEN and SFGEN Commands Now Support Nodal Components

The **CNTGEN** (contact generation) and **SFCGEN** (surface generation) commands have been updated to support nodal components in addition to the previously supported EFACE (element face) components. Nodal components are supported for both the 2D and 3D contact elements to setup contact and MPC (multi-point constraint) surface constraints.

If nodal components are provided for the contact and target surfaces, issuing **CNTGEN** generates contact and target elements based on the dimensionality of the underlying mesh elements:

For 2D Plane Elements: CONTA172 contact elements and TARGE169 target elements.

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

For 3D Solid/Shell, Gasket, and Interface Elements: CONTA174 contact elements and TARGE170 target elements.

The **SFCGEN** command now creates MPC surface-based constraints over the surfaces formed by the nodal components based on the user-defined contact element type (ICTY).

For 2D Plane Elements: CONTA172 contact elements with pilot nodes meshed by TARGE169 target nodes (**TSHAPE**,PILO).

For 3D Solid/Shell, Gasket, and Interface Elements: CONTA174 contact elements associated with a pilot node meshed by TARGE170 (**TSHAPE**,PILO).

1.1.6. Performance Improvements for Multiple Spot Welds

Overall performance has been improved for the generation of multiple spot welds with node-to-surface configuration. Modeling large assemblies for the automotive industry requires generating several layered spot welds at multiple locations. Issuing several spot weld generation (**SWGEN**) commands to simulate this process can be computationally expensive.

To reduce spot weld generation time of large assemblies, memory optimization and the definition of element attributes like real constants, are now done once at **SWGEN**,START. The program automatically assigns the next available real constant for subsequent spot welds, speeding up the overall process.

1.1.7. 2D-3D Bonded MPC Contact for Solid Assemblies Supports Thermal and Structure-thermal Coupling

The 2D-to-3D bonded multi-point constratint (MPC) contact feature for solid assemblies now supports thermal and coupled structural-thermal analyses. You can combine multiphysics 2D elements (PLANE222, PLANE223, PLANE292, PLANE293) elements with 3D elements (SOLID278, SOLID279, SOLID226, SOLID227) to simulate heat transfer and deformation due to temperature change across their bonded contact interfaces. For more information, see Modeling a 2D/2.5D-3D Solid Assembly in the *Contact Technology Guide*.

1.1.8. DMP Performance Improvement of MPC CONTA175 Element

To boost the performance of large-scale distributed-memory parallel (DMP) contact models, the solver now separates the underlying elements of CONTA175 into different processing domains. Previously the CONTA175 elements and their underlying elements were grouped into the same DMP domain. This change activates when the CONTA175 elements do not need underlying element information (KEYOPT(5) = 0, KEYOPT(11) = 0, KEYOPT(4) = 0 or 3, target element TARGE170 KEYOPT(7) is not equal to 0, and the contact pair does not include geometry correction). This performance improvement results in a lower load balance ratio and better scalability for high-core DMP models involving contact.

1.1.9. Improved Convergence when Simulating Assemblies that Include Gasket Elements

The initial contact stiffness for the contact element on top of a gasket element has been updated to improve convergence behavior for nonlinear structural analysis.

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

1.2. Elements and Nonlinear Technology

Release 2026 R1 includes the following enhancements to elements and nonlinear technology used in structural analyses:

1.2.1. Contact-based Displacement Increment Control

1.2.2. Updates to Smart Stop Criterion

1.2.3. Gasket Element Improvement

1.2.4. Direct Element Embedding Enhancement

1.2.5. General Axisymmetric Element Enhancements

1.2.6. Updates to 3D Structural Shell Elements

1.2.7. Extended support for Nonlocal Damage Formulation

1.2.8. New Standard Variant for Void Creation in Generalized Remeshing for Adaptive Nonlinear Damage (GRAND)

1.2.9. Automatic Identification and Removal of Orphaned Element Patches in Nonlinear Adaptivity

1.2.10. Support for Arruda-Boyce, Bergstrom-Boyce and TNM Material Models in Nonlinear Adaptivity

1.2.1. Contact-based Displacement Increment Control

You can now control the displacement increment based on the pinball region of open contact pairs. Issue **CUTCONTROL**, DSPLIMIT, CONT to limit the substep incremental displacement to a factor times the minimum pinball radius of all open contact pairs. This helps prevent missing the contact when a displacement increment causes a contact pair to transition from no-contact to far-field. See the **CUTCONTROL** command description for details.

1.2.2. Updates to Smart Stop Criterion

An optional stop tolerance (*STOP_TOL*) argument is available for the **NLHIST** command's smart stop feature. The *STOP_VALUE*, *STOP_COND* and *STOP_TOL* arguments automatically tailor a nonlinear solution to stop when the value of a tracked variable reaches the specified range, using bisection followed by time increment prediction.

1.2.3. Gasket Element Improvement

A new algorithm is used for updating the user-defined thickness of 3D gasket elements (INTER194 and INTER195) with the thin-solid option (KEYOPT(2) = 2 or 3) in geometric non-linear analyses (**NLGEOM** ,ON). The new algorithm improves the non-linear solution robustness and the accuracy of element results.

1.2.4. Direct Element Embedding Enhancement

The SHELL229 coupled-field shell elements are now supported as embedded elements, and 3D coupled-field solid elements (SOLID225, SOLID226 and SOLID227) are now supported as base elements in the direct-embedding workflow. For more information, see Direct-Embedding Workflow.

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

1.2.5. General Axisymmetric Element Enhancements

The general axisymmetric elements SOLID272 and SOLID273 now support the mono harmonic option (KEYOPT(2) = 1 and **MODE**) in linear perturbation modal analyses, enabling the analysis of higher-frequency modes. Furthermore, by changing the **MODE** number, multiple load steps are supported in subsequent modal analyses. The out-of-plane contact behavior for SOLID272 and SOLID273 (KEYOPT(2) = 1) is supported using a 3D contact pair: CONTA175 and TARGE170.

1.2.6. Updates to 3D Structural Shell Elements

The 3D structural shell elements (SHELL181 and SHELL281) now support selected solution output locations using the **EMSEL** command. This feature provides flexibility in selecting output locations.

1.2.7. Extended support for Nonlocal Damage Formulation

PLANE183 now supports the nonlocal damage formulation. SOLID186 and SOLID187 elements have been updated to support nonlocal damage when using the mixed u–p formulation. These enhancements enable use of generalized fatigue damage (implicit gradient regularization) to better capture the distributed nature of damage in real materials by averaging variables over a finite neighborhood using a characteristic length scale. This leads to more objective and reliable simulation results and improved convergence stability. Also, results no longer spuriously depend on mesh size, especially when material softening or strain localization occurs.

1.2.8. New Standard Variant for Void Creation in Generalized Remeshing for Adaptive Nonlinear Damage (GRAND)

Multiple voids per substep can be created based on a criterion selection, and these voids are non-overlapping. The damaged element with the maximum error indicator value is selected as the element for single void insertion, using the **NLADAPTIVE**,,,REMMAT command with *VAL3* = 1. For the multiple voids per substep approach, you can include loaded boundary elements by issuing the **NLMESH**,VBND command with *VAL1* = 1. With this new variant, you can now create more than one void per substep based on the criterion value and all damaged elements are considered for void creation.

1.2.9. Automatic Identification and Removal of Orphaned Element Patches in Nonlinear Adaptivity

Automatic detection and deletion of orphaned element patches after the element removal process (**NLADAPTIVE**,,,REMELM) are now supported using the **NLMESH** command with *Control* = ISLD. This leads to a more stable solution process and convergence when many elements are deleted.

1.2.10. Support for Arruda-Boyce, Bergstrom-Boyce and TNM Material Models in Nonlinear Adaptivity

The Arruda-Boyce, Bergstrom-Boyce and the TNM material models are now supported in a nonlinear adaptivity analysis and rezoning analysis. For more information, see Nonlinear Mesh Adaptivity Requirements and Limitations and Rezoning Requirements and Limitations.

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

1.3. Material and Fracture Modeling

Release 2026 R1 includes the following enhancements to material modeling and fracture analysis technology used in structural analyses:

1.3.1. Finite-strain Material Models Enhancement
1.3.2. Material Curve-Fitting Enhancements
1.3.3. Support for Anisothermal Derivatives in Thermal Analyses
1.3.4. Damage Cap Enhancements
1.3.5. Improved UserCreep Handling in the Extended Drucker-Prager Cap Creep Model
1.3.6. Advanced Grain-Growth Model for Sintering
1.3.7. New Sintering Stress Evolution Models
1.3.8. Conversion of Initial Stress to Crack-surface Traction in Multiple Loadsteps
1.3.9. Enhancement of SIF (Stress-Intensity Factors) Calculation
1.3.10. SMART Support with Substructuring
1.3.11. New Fracture Benchmark Problem
1.3.12. Support of Mixed Element Types in SMART Crack-Growth Analysis

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. Finite-strain Material Models Enhancement

Finite-strain material models now support swelling effects that can act independently of or in combination with thermal deformation. The total deformation gradient is multiplicatively split into mechanical, thermal, and swelling parts, with stresses driven by the mechanical component.

1.3.2. Material Curve-Fitting Enhancements

AI-based automatic initialization of material parameters is now supported for more plasticity models, including both rate-independent and rate-dependent plasticity. For more details, see Plasticity Models in Step 4. Initialize the Coefficients.

You can now fix selected coefficients prior to automatic initialization, so that only unfixed coefficients are initialized. This feature is supported for hyperelasticity and plasticity.

You can also now use automatic fitting in hyperelasticity to generate material parameters directly from experimental data, skipping the curve fitting process. This is done with the **TBFT** command. For details, refer to Automatic Fitting.

1.3.3. Support for Anisothermal Derivatives in Thermal Analyses

The program can now optionally consider anisothermal derivative terms in stiffness and damping matrices for temperature dependent material properties in a thermal analysis with and without nonlocal damage. This can help with convergence when material parameters are strongly temperature dependent.

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

1.3.4. Damage Cap Enhancements

A damage cap can now optionally be applied to restrict further damage evolution beyond a specified limit for Regularized Generalized Damage and Ductile Damage. This can prevent elements from reaching unrealistically low stiffness values, which could lead to convergence difficulties.

1.3.5. Improved UserCreep Handling in the Extended Drucker-Prager Cap Creep Model

The extended Drucker-Prager Cap Creep model can now be used with custom creep models (**TB**,CREEP,,,,100) implemented using the UserCreep subroutine. You can now use the `get_ElmData` subroutine with the 'ECCR' option in UserCreep to identify if the creep model for the compaction cap region or the shear region is requested.

1.3.6. Advanced Grain-Growth Model for Sintering

A new general power law model for grain-growth kinetics is now available, enabling more accurate simulation of normal grain growth during sintering. Unlike the previous parabolic model, this approach supports variable growth exponents, making it suitable for diverse materials such as ceramics and metals used in powder metallurgy.

1.3.7. New Sintering Stress Evolution Models

Three new stress evolution models are available for accurate modeling of sintering stress. This enhancement builds upon existing stress calculation by adjusting the calculated stress. The new models are based on work by McMeeking & Kuhn, Ashby, and Shinagawa.

1.3.8. Conversion of Initial Stress to Crack-surface Traction in Multiple Loadsteps

For SMART crack-growth analysis, you can now change the mesh-independent initial-stress data at any loadstep for conversion to an equivalent crack-surface traction load. The existing data must be cleared from the solution database before reading the new data. By default, the initial-stress data specified at the first loadstep is carried over to subsequent loadsteps.

When this load method is used with crack-initiation, the traction load is applied only after the crack begins to form and its surfaces become available for load application.

1.3.9. Enhancement of SIF (Stress-Intensity Factors) Calculation

The fracture parameter SIF calculation can now provide accurate results for both 2D problems (PLANE182 and PLANE183) and 3D problems (SOLID185 and SOLID186) undergoing large deflection and/or rotation.

1.3.10. SMART Support with Substructuring

When a model includes any superelement (MATRIX50), SMART crack growth can now be analyzed in the nonsuperelement mesh. The superelement must not be in the crack growth remeshing region. The superelement should be connected to the nonsuperelements via shared nodes or constraint equations (**CE** or **CEINTF**) or contact surface elements (CONTA174).

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

1.3.11. New Fracture Benchmark Problem

The following benchmark problem has been added at this release:

- Center-Crack Specimen Subjected to Residual Stress in the *Fracture Analysis Guide*

1.3.12. Support of Mixed Element Types in SMART Crack-Growth Analysis

SMART crack-growth analysis now supports tetrahedral-to-hexahedral models. This feature now allows you to use SOLID185 and SOLID186 elements mixed with SOLID187 elements. SOLID185 and SOLID186 elements can be in wedge, hex, or pyramid shapes. After crack growth occurs, the remeshing domain mainly consists of SOLID187 elements. If a high-order element is connected to a low-order element, the remesher will use a special connection by dropping the mid-side nodes in the element.

1.4. Linear Dynamics

Release 2026 R1 includes the following enhancements for structural analyses involving linear dynamics:

1.4.1. Enhanced Restart Capability for Harmonic Balance Method Analysis

1.4.2. Multistage Cyclic Symmetry Supports Transient Thermal Analysis

1.4.3. Joints with Cyclic Symmetry Linear Perturbation Harmonic Analyses

1.4.1. Enhanced Restart Capability for Harmonic Balance Method Analysis

In previous versions, restarting a Harmonic Balance Method (HBM) analysis required manually extracting the initial guess from a results file (`.rst`) using a custom macro. This approach often led to discontinuities in the generated results files and added complexity to the workflow.

You can now restart an HBM analysis using the standard command **ANTYPE**,HARMIC,RESTART, which includes the option to select the desired substep. The results files are rewritten to maintain solution continuity in the selected frequency range. For details, see Restarting an HBM Analysis in the *Harmonic Balance Method Analysis Guide*.

1.4.2. Multistage Cyclic Symmetry Supports Transient Thermal Analysis

Transient (**ANTYPE**, TRANS) thermal multistage cyclic symmetry analysis is now supported in addition to the already supported multistage steady-state (**ANTYPE**, STATIC) thermal analyses. Models supported for these analyses have only temperature degrees of freedom (DOFs) active during the solution. See Multistage Thermal Analysis in the *Multistage Cyclic Symmetry Analysis Guide*.

1.4.3. Joints with Cyclic Symmetry Linear Perturbation Harmonic Analyses

Cyclic symmetry linear perturbation harmonic analysis now supports the use of MPC184 joint elements in the model.

2. Multiphysics

Release 2026 R1 includes the following enhancements for analyses involving multiphysics environments:

2.1. Acoustics

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

2.2. Thermal
2.3. Coupled-Field

2.1. Acoustics

The following enhancements for acoustic analysis are available in this release:

2.1.1. Aerodynamic Source Terms (Lighthill's Analogy) in the Wave Equation
2.1.2. Perfectly Matched Layer (PML) for Axially Symmetric Elements in Transient Analysis
2.1.3. Logarithmic absorption function option for Perfectly Matched Layer (PML) and Irregular Perfectly Matched Layer (IPML) acoustic elements

2.1.1. Aerodynamic Source Terms (Lighthill's Analogy) in the Wave Equation

Aerodynamic sources defined in terms of the divergence of Lighthill's tensor are now supported for 3D acoustic fluid elements (FLUID30, FLUID220, and FLUID221) and 2D acoustic fluid elements (FLUID243 and FLUID244) in harmonic acoustic analyses. With this new feature, you can use flow sources obtained from Computational Fluid Dynamics (CFD) simulations to compute broadband noise generated by turbulent flows. For details, see Lighthill's Analogy for a Co-rotating Vortex Pair and Aerodynamic Source (Divergence of Lighthill's Tensor) in the *Acoustic Analysis Guide*.

2.1.2. Perfectly Matched Layer (PML) for Axially Symmetric Elements in Transient Analysis

The 2D acoustic fluid elements, FLUID243 and FLUID244, now support the axisymmetric PML option for transient acoustic simulations.

2.1.3. Logarithmic absorption function option for Perfectly Matched Layer (PML) and Irregular Perfectly Matched Layer (IPML) acoustic elements

A new logarithmic absorption function option is available for acoustic PML and IPML elements. This option enhances wave attenuation and minimizes reflections, particularly when the available radiation space is limited. Compared to conventional polynomial absorption profiles, the logarithmic function provides smoother impedance matching and improved accuracy in harmonic acoustic analyses. For details, see the **PMLOPT** command description and Absorption function for IPML and PML in the *Theory Reference*.

2.2. Thermal

The following enhancements for thermal analysis are available in this release:

2.2.1. New NMISC Values for Performing Energy Balance
2.2.2. View Factor Sums for Radiation Surface Facets

2.2.1. New NMISC Values for Performing Energy Balance

New element NMISC quantities have been added to the current technology thermal elements (PLANE292, PLANE293, SOLID278, SOLID279, SOLID291, and SHELL294). Use them to easily perform

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

an energy balance in postprocessing of a transient or steady-state thermal analysis. See element descriptions for details.

2.2.2. View Factor Sums for Radiation Surface Facets

When using the radiosity solver for thermal radiation problems, the view factor sum for each surface facet (SURF251 or SURF252) is now available (NMISC,11) to help you identify which radiation surfaces are dominant and how they affect others. See the element descriptions for details.

2.3. Coupled-Field

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

2.3.1. Expanded Capability of User-Programmable Features

2.3.2. SHELL229 Supports Structural-thermal and Structural-thermoelectric Analyses

2.3.3. Extended Support for Nonlocal Damage Formulation

2.3.1. Expanded Capability of User-Programmable Features

Newly supported analyses: User-programmable features (UPFs) for coupled-field elements (PLANE222, PLANE223, SOLID225, SOLID226, and SOLID227) support more analyses, including structural-thermal, thermal-electric, structural-thermal-electric, and piezoelectric. You can now customize material models, loads, constitutive equations, element matrices, and element results in the above analyses.

New Customizable Quantities: The scope of customizable quantities has been extended to include diffusion strain, dielectric permittivity, electric flux density, complex stiffness and mass matrices, complex load vector, and state variables shared with UserMat and UserMatTh subroutines.

Additional utility functions: New utility functions have been added to conveniently access nodal and element results during element customization. Examples include velocities, accelerations, complex displacements, element principal and equivalent strains and stresses, as well as strain and stress intensity.

2.3.2. SHELL229 Supports Structural-thermal and Structural-thermoelectric Analyses

SHELL229 is a current-technology shell element with coupled-field capabilities that previously supported thermo-electric analysis. It now supports structural-thermal and structural-thermoelectric analyses as well. The element has up to 8 degrees of freedom at each node, including temperature and voltage in membrane format, as well as 3D translation and rotation for structural analysis. See VM174 and VM223 in the *Ansys Mechanical APDL Verification Manual*.

2.3.3. Extended Support for Nonlocal Damage Formulation

Nonlocal damage formulation is now enabled for PLANE223 for structural–thermal, structural–diffusion, and structural–thermal–diffusion analyses. SOLID226 and SOLID227 elements have been updated to support nonlocal damage when using the mixed u-P formulation (KEYOPT(11) = 1). These enhancements enable use of generalized fatigue damage (implicit gradient regularization) to better capture the distributed nature of damage in real materials by averaging variables over a finite

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

neighborhood using a characteristic length scale. This leads to more objective and reliable simulation results and improved convergence stability. Also, results no longer spuriously depend on mesh size, especially when material softening or strain localization occurs.

3. Solvers

Release 2026 R1 includes the following improvements to the solution process:

3.1. Resource Prediction Feature

3.2. PCG Block Lanczos Eigensolver

3.3. PCG Solver Enhancements

3.1. Resource Prediction Feature

The new **PREA** command introduces resource prediction, a feature that estimates computational resource requirements before solving. This helps identify the optimal hardware environment for simulations.

By analyzing the current load step, analysis type, solver type, and core count, **PREA** provides estimates for:

- Solution elapsed time
- Total memory usage
- Total disk space usage
- GPU recommendation

These predictions support distributed-memory parallel (DMP) solutions and assist with performance and resource allocation. While predictions offer valuable guidance, they do not guarantee solution success or solver convergence.

3.2. PCG Block Lanczos Eigensolver

The preconditioned conjugate gradient (PCG) Block Lanczos eigensolver is now officially released.

This eigensolver is a variant of the Block Lanczos method designed for large symmetric eigenvalue problems in modal analyses. It uses the same automated shift strategy as the standard Block Lanczos method but adds PCG iterative solver technology alongside the existing sparse direct solver.

To use this eigensolver, combine the Block Lanczos method (**MODOPT**,LANB) with the PCG solver (**EQSLV**,PCG). This combination automatically activates the new eigensolver, which improves performance and memory efficiency.

For more information, see the **MODOPT** and **EQSLV** commands.

3.3. PCG Solver Enhancements

The PCG solver requires less memory than in previous releases to solve models when:

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

- Running on higher core counts (more than 64 cores).
- Constraint equations and/or coupling equations exist in the model.

4. High-Performance Computing

Release 2026 R1 includes the following improvements to high-performance computing (HPC):

4.1. Distributed-Memory Parallel Processing Enhancements
4.2. GPU Acceleration Feature: New Support and Updates
4.3. Recommended Devices for the GPU Accelerator: New and Updated GPUs

4.1. Distributed-Memory Parallel Processing Enhancements

Enhancements include:

- The default support for message passing interface (MPI) software has been updated to Intel MPI 2021.16.1 for both Linux and Windows. Previously, the default was Intel MPI 2021.14.2.
- When using AMD processors, the default support for message passing interface (MPI) software has been updated to Open MPI 4.1.8. Previously, the default was Open MPI 4.0.5.

4.2. GPU Acceleration Feature: New Support and Updates

New Support

- Windows now supports ROCm libraries version 6.4.2.

Updates

- For Linux, the ROCm libraries have been updated to 7.0.2. The previous release used version 6.3.3.
- For Windows and Linux, the CUDA libraries have been updated to 13.0.2. The previous release used version 12.8.
- The recommended GPU cards table now includes detailed specifications on performance metrics, memory size, and bandwidth.

Performance Improvements

- The performance of the PCG solver has improved with GPU acceleration, as the Mechanical APDL application now dynamically offloads as many equation solver calculations as possible onto the GPU. This enables the solver iterations to run entirely on the GPU with minimal CPU communication.

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

4.3. Recommended Devices for the GPU Accelerator: New and Updated GPUs

New Feature

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

- The Mechanical APDL application now supports GPU acceleration on Windows for AMD GPUs.

Added Cards

The following cards have been tested and added to the list of recommended GPU devices for the GPU accelerator:

- **Windows:**
  - AMD Radeon AI PRO R9700
  - AMD Ryzen AI Max+ PRO 395
  - AMD Ryzen AI Max+ 395
  - AMD Radeon PRO W7900
  - NVIDIA GeForce RTX 5090
  - NVIDIA GeForce RTX 5080
  - NVIDIA GeForce RTX 5070 Ti
- **Linux:**
  - AMD Instinct MI355X
  - NVIDIA GeForce RTX 5090
  - NVIDIA GeForce RTX 5080
  - NVIDIA GeForce RTX 5070 Ti

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

5. Other Enhancements

The following other enhancements have been added:

5.1. Compiler Update for User-Programmable Features on Linux Systems
5.2. New Coded Database File Command
5.3. Modified Coded Database File Command
5.4. Expanded Element Face Component Creation Options

5.1. Compiler Update for User-Programmable Features on Linux Systems

The supported GNU GCC compiler for user-programmable features has been updated to version 12.3.0. The previous release used version 8.2.0.

For more information, see Compiler Requirements for Linux Systems in the *Ansys, Inc. Installation Guides* and Compiling and Linking UPFs on Linux Systems in the *Programmer's Reference*.

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

5.2. New Coded Database File Command

The following new coded database (CDB) file command is available:

- NSMBLOCK – Defines data for the non-structural mass of elements computed by the **NSMASS** command.

5.3. Modified Coded Database File Command

The following CDB file command has been enhanced or otherwise modified:

- /BLOCK – Signals the start and end of block input. The previous *Control* option has been replaced by two parameters—*Parm1* and *Parm2*—to support more efficient handling of element blocks.

5.4. Expanded Element Face Component Creation Options

You can now construct element face (EFACE) components using:

- Named element (ELEM) components defined with the **CM**, CMBLOCK, or EBLOCK (with the **APPCM** agrument) commands.
- Assemblies of ELEM components, including up to three levels of nested assemblies.

When using component names or assemblies to create EFACE components, compact notation is not supported.

6. Commands

This section describes changes to commands at Release 2026 R1:

6.1. New Commands
6.2. Modified Commands
6.3. 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.

6.1. New Commands

The following commands are new for this release.

- **FGRAD** – Directs the solver to include the effects of field-dependent gradients in the solution.
- **PREA** – Activates resource prediction for supported analysis and solver types.
- **WKSP** – Automatically adds weak springs at the nodes of degrees of freedom with small or near-zero pivots during solution.

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

6.2. Modified Commands

The following commands have been enhanced or otherwise modified:

- **AMENV** – Specifies the build-environment thermal boundary conditions in an additive manufacturing (AM) analysis. New options (*RADOPT*, *EMIS*, and *SBC*) enable you to define radiation boundary conditions in a powder bed fusion (PBF) simulation.
- **BF** – Defines a nodal body-force load. A new DIVT label type enables the use of the divergence of Lighthill's tensor as a body-force excitation source.
- **CNCHECK** – Provides and/or adjusts the initial status of contact pairs. Use the new label MPC for *InterType* when *Option* = MERGE (**CNCHECK**,MERGE,,,,MPC) to merge MPC bonded contact pairs that share the same nodes or elements.
- **CUTCONTROL** – Controls time-step cutback during a nonlinear solution. Use the new label CONT for *Value* to limit the substep incremental displacement based on the minimum pinball radius of all open contact pairs and help ensure accurate contact detection.
- **EMSEL** – Specifies conditions for selecting reinforcing members or shell layers. The pre-processing and post-processing commands (**PLESOL**, **PLNSOL**, **PRESOL**, **PRNSOL**, **ETABLE** and **EPLOT**) now support selected layered results for shell sections and elements (SHELL181 and SHELL281).
- ***GET** – Retrieves a value and stores it as a scalar parameter or part of an array parameter. Use the new preprocessing data type output (*IT1NUM* = BSTF when Entity = SHEL) to retrieve the bending stiffness scale factor.
- **MONITOR** – Controls contents of variable fields in the nonlinear solution monitor file. An additional field has been inserted between variable fields 1 (*VARIAB 1*) and 2 (*VARIAB 2*) to report current memory usage (in GB) at each converged substep. For more information, see Change in Format of Nonlinear Solution Monitor File (.mntr) (p. 74).
- **NLADAPTIVE** – Defines the criteria under which the mesh is refined or modified during a nonlinear solution. For *Option* = ESTN and *Option* = PSTN:
  - *VAL3* is now used as a flag for variant of void creation.
  - *VAL6* is now unused.
- **NLHIST** – Specify results items to track during solution. Smart stop criterion now includes an optional stop tolerance (*STOP_TOL*) that automatically stops the solution when a tracked variable is within the specified percentage tolerance of the desired value (*STOP_VALUE*). *STOP_TOL* defaults to 0.025 (2.5%).
- **NLMESH** – Controls remeshing in nonlinear adaptivity. Use the new *Control* option, VBND, to include or exclude loaded boundary elements for void creation in multiple voids per substep variant (**NLADAPTIVE**,,,REMMAT). The new *Control* option, ISLD, allows you to remove all orphaned elements or element patches (islands) after an element removal operation (**NLADAPTIVE**,,,REMELM).
- **PMLOPT** – Defines perfectly matched layers (PMLs) or irregular perfectly matched layers (IPML). A Logarithmic absorption function is now supported in addition to the polynomial one. To select the logarithmic function, which is recommended for analyses with a radiation space that is less than

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

one wavelength, specify the new NF option for *WOptXm*, *WOptXp*, *WOptYm*, *WOptYp*, *WOptZm*, *WOptZp* as desired on the **PMLOPT** command.

- **PIVCHECK** – Controls the behavior of an analysis when a negative or zero equation solver pivot value is encountered. The existing *Key* = AUTO option now prints the list of pivot values along with their corresponding degrees of freedom, in addition to the error or warning message.
- **SECMODIF** – Modifies section properties based on the keyword. You can now modify the properties of shell, beam, pipe, link, and reinforcing sections. Selected layer modification is also supported for shell sections using the **EMSEL** command.
- **SOLVE** – Starts a solution. A new option, **SOLVE**,ELFORM,,REFORM, has been added to trigger multiple load steps in linear perturbation modal analysis using SOLID272 and SOLID273.
- **TB** – Activates a data table for linear and nonlinear material properties or special element input.
  - Poisson's ratio for automatic fitting is now supported using **TB**,EXPE with *TBOPT* = POIS.
  - Damage cap for regularized generalized damage approach is now supported using **TB**,CDM with *TBOPT* = DCAP.
  - The inelastic dissipation rate fraction and latent heat coefficients are now supported using **TB**,THERM with *TBOPT* = DISS and *TBOPT* = LATH, respectively.
- **TBFT** – Performs material curve-fitting operations. Use the new option, *Oper* = AFIT, to automatically read and fit experimental data from **TB**,EXPE and save material model to the **TB** command format. Use the new option, *Oper* = AFPAR, to enable or disable diagnostics during the automatic fitting process.

6.3. 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.

7. Elements

This section describes changes to elements at Release 2026 R1:

7.1. New Elements
7.2. Modified Elements
7.3. Undocumented Elements

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

7.1. New Elements

There are no new elements this release.

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

7.2. Modified Elements

The following elements have been enhanced:

- PLANE222, PLANE223, SOLID225, SOLID226, SOLID227 - these elements now support user-programmable features (KEYOPT(12) = 1) with structural-thermal, thermal-electric, structural-thermoelectric, and piezoelectric analyses.
- INTER194 and INTER195 – In combination with a user-defined thickness (KEYOPT(7) = 1) and the thin-solid option (KEYOPT(2) = 2 or 3), a new algorithm is used for updating the user-defined thickness in geometric nonlinear analyses (**NLGEOM** ,ON). The new algorithm improves the nonlinear solution robustness and the accuracy of element results.
- SHELL229 – This current-technology shell element with coupled-field capabilities now supports structural-thermal and structural-thermoelectric analyses.
- SOLID272 and SOLID273 (KEYOPT(2) = 1 and **MODE**) – The mono harmonic option is now supported in linear perturbation modal analyses.
  - Subsequent modal analyses now support the **MODE** command, enabling the analysis of higher-frequency modes within a linear perturbation modal analysis.
  - Multiple load steps with **MODE** number changes are now supported in subsequent modal analyses.
  - Expanded 3D plots of mode shapes are now supported using KEYOPT(3) = 1 and **/ESHAPE**,1.
- SOLID272 and SOLID273 (KEYOPT(2) = 1) – The out-of-plane contact behavior is now supported using a 3D contact pair: CONTA175 and TARGE170.
- SURF159 – This general axisymmetric surface element now supports the mono harmonic option of SOLID272 and SOLID273 (KEYOPT(2) = 1) using the **MODE** command.
- SURF251 and SURF252 – These radiosity surface elements now output the view factor sum via NMISC,11. You can use the view factor sum for each surface facet to identify the dominant radiation surfaces and evaluate how they affect others.
- PLANE292, PLANE293, SOLID278, SOLID279, SOLID291, SHELL294 – New element quantities (NMISC) useful for performing an energy balance at each substep are available. They provide the element quantity of each term in the energy balance equation. For each converged substep in **/POST1**, you can read these element quantities using the **ETABLE** command and sum them via the **SSUM** command to give the overall contributions for all elements in the model.
- PLANE183, PLANE223 – These elements now support the use of generalized fatigue damage (implicit gradient regularization), which better captures the distributed nature of damage in real materials. This leads to the following key benefits: objective and reliable results, mesh insensitivity (especially when material softening or strain localization occurs), and improved convergence stability.
- SOLID186, SOLID187, SOLID226, SOLID227 – These elements now support the use of generalized fatigue damage (implicit gradient regularization) when using the mixed u-P formulation (KEYOPT(11) = 1). The use of generalized damage can better capture the distributed nature of damage in real materials.

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

- SOLID186 – This element type now supports single-value results. For more information, see Solution Output in the *Element Reference*.

7.3. 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.

8. Documentation

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

8.1. Cyclic Symmetry Documentation Examples

The cyclic symmetry example problems were updated to improve readability and ease of use (see Example Cyclic Symmetry Analyses in the *Cyclic Symmetry Analysis Guide*).

8.2. 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.

8.3. *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. 77) in the Update Guide section of this document.

9. 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.

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

There are no new example problems this release.

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*.

10. Mechanical APDL Release 2026 R1 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 2026 R1 product-update information presented by the Mechanical APDL development and testing teams:

10.1. Backward Compatibility
10.2. Feature Updates Causing Result or Behavior Changes
10.3. Known Incompatibilities
10.4. Known Issues
10.5. Resolved Issues and Defects
10.6. 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.

10.1. Backward Compatibility

Mechanical APDL Release 2026 R1 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.

10.2. Feature Updates Causing Result or Behavior Changes

The following Mechanical APDL feature updates in Release 2026 R1 are known to produce program behaviors or analysis results that differ from those of the prior release:

10.2.1. Change in .ldhi Files
10.2.2. Changes in Automatic Hybrid Parallel
10.2.3. Change in Internal Multipoint Constraint (MPC) Reporting in .ldhi and .cnd Files
10.2.4. Moment Convergence Check at Pilot Node of Target Elements with Rotational DOFs
10.2.5. Harmonic Balance Method Uses Sparse Solver by Default
10.2.6. Output Displays Elapsed Time
10.2.7. Change in Format of Nonlinear Solution Monitor File (.mntr)
10.2.8. Restart Mechanism Improves Conjugate Gradient (CG) Solver Robustness

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

10.2.9. Change in Preconditioned Conjugate Gradient (PCG) Lanczos Eigensolver
10.2.10. Output behavior of mapdl.run in a *PYTHON block
10.2.11. Change in Impedance Sheet Formulation for Acoustic Analysis
10.2.12. Change in Default for Ansys Product Improvement Program (APIP)
10.2.13. Change in Behavior Due to Backstress Scaling in the Chaboche Model for Non-Isothermal Loading

10.2.1. Change in `.ldhi` Files

The Mechanical APDL application no longer writes duplicated tabular load data across load steps to `*.ldhi` files when the data remains unchanged. This reduces file size and improves efficiency. During restarts, the application reads the necessary data from the `*.rdb` file, ensuring continuity without requiring repeated table entries. In previous releases, duplicated tables were written to `*.ldhi` files in every load step.

10.2.2. Changes in Automatic Hybrid Parallel

The automatic hybrid parallel feature, also known as auto-hybrid, now prevents hardware oversubscription on large clusters that previously degraded performance. The following behavior changes are included:

Single-node runs at low core counts

Auto-hybrid now applies threads only to large domains when reducing core counts (for example, `-np 12` becomes `-np 4` with 8 threads). This change may alter the solution printout compared to previous releases. Previously, threads were distributed evenly across all domains, regardless of size.

Multi-node runs

Auto-hybrid activates only when all compute nodes have the same number of CPU cores (homogeneous configuration). Therefore, the number of domains and threads determined by auto-hybrid parallel heuristics may differ from previous releases. This change helps prevent hardware oversubscription and associated performance decline.

Oversubscription protection

A final configuration check now runs before execution. If oversubscription is detected, the program issues a fatal error and stops the run. To proceed, rerun the solution with the `-nt 1` command-line argument, which disables the auto-hybrid feature. Previously, no such check existed, and oversubscription could lead to poor performance or instability.

Restart and downstream analysis behavior

When restarting or continuing an analysis, auto-hybrid restores the previous configuration of processes and threads. If a downstream analysis launches with more processes than the upstream analysis, auto-hybrid reduces the number of active processes to match the upstream configuration.

For example:

- Upstream modal analysis runs with `-np 4`.

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

- Downstream harmonic analysis launches with `-np 8`.

In previous releases, the harmonic analysis would reduce to `-np 2 -nt 2`. In Release 2026 R1, the harmonic analysis reduces to `-np 2` and does not activate additional threads (`-nt 1`).

Recommendation: Launch downstream analyses with the same or fewer processes than the upstream analysis to avoid unexpected reductions.

10.2.3. Change in Internal Multipoint Constraint (MPC) Reporting in .ldhi and .cnd Files

MPC (KEYOPT(2) = 2) bonded (KEYOPT(12) =5 or 6) contact excluding initial geometrical penetration or gap and offset (KEYOPT(9) = 1) no longer reports PENE, GAP, and SLIDE in the element nodal solution, `*.ldhi` and `*.cnd` files.

10.2.4. Moment Convergence Check at Pilot Node of Target Elements with Rotational DOFs

Moment convergence is now checked at the pilot node of TARGE170 when target elements have rotational degrees of freedom (DOFs), improving accuracy. Previously, this check at the pilot nodes was ignored if there were no structural elements with rotational DOFs in the model.

10.2.5. Harmonic Balance Method Uses Sparse Solver by Default

The sparse solver is now used by default in harmonic balance method (HBM) analyses. This change is expected to improve overall performance, particularly for systems with many degrees of freedom. While the solver may introduce slight numerical differences in results, these are generally negligible and should not affect the accuracy or reliability of the analysis.

10.2.6. Output Displays Elapsed Time

Output messages now report the **elapsed time** (wall clock time) since the start of the program, instead of CPU time. This change provides a more accurate and relevant measure of total runtime. It accounts for all delays, including those outside CPU execution, and avoids confusion caused by thread-level CPU time aggregation.

Example output:

```
***** ROUTINE COMPLETED *****  ELAPSED TIME =        3.461
*** NOTE ***                   ELAPSED TIME =       12.982   TIME= 08:00:00
*** WARNING ***                ELAPSED TIME =       11.432   TIME= 09:00:00
*** WARNING ***                ELAPSED TIME =       65.347   TIME= 10:00:00
```

In previous releases, the output used the label **CP** to indicate CPU time. It measured only the time the processor spent executing instructions, excluding time spent on I/O operations, network delays, GPU acceleration, and other non-CPU activities. Additionally, in shared memory parallel (SMP) mode, CPU time was summed across all threads and cores. This could result in misleadingly high values, making it appear that certain simulation steps took longer than they did.

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

10.2.7. Change in Format of Nonlinear Solution Monitor File (`.mntr`)

The nonlinear solution monitor file (`*.mntr`) now includes an additional field between variable fields 1 (*VARIAB 1*) and 2 (*VARIAB 2*). This new field reports current memory usage (in GB) at each converged substep, enabling better tracking of memory consumption during nonlinear analysis.

In previous releases, the `*.mntr` file contained variable fields 1 through 4. With this change, fields 2 through 4 have shifted to become variable fields 3 through 5.

For more information, see **MONITOR**.

10.2.8. Restart Mechanism Improves Conjugate Gradient (CG) Solver Robustness

This release introduces a restart mechanism for all CG drivers to improve solver reliability. The enhancement may improve solutions for models that previously converged slowly (below 1.0E-6) or failed to converge. Some test cases may show a slight increase in iteration counts due to the additional cycle.

When the restart mechanism is triggered, the solver performs an extra cycle of iterations. This second cycle starts from the most accurate solution found in the first cycle. Both cycles use the same maximum number of iterations.

Additional behavior changes includes:

- For linear systems with low condition number estimates and relaxed solver tolerances, the CG algorithm locally reduces the maximum number of iterations per cycle.
- During the first cycle, if the solution diverges or approaches within three orders of magnitude of the requested tolerance, the solver aborts the cycle and starts a new one.

In previous releases, when the solver reached the maximum number of iterations without meeting the tolerance, it would error out without completing the solution.

For more information, see Conjugate Gradient (CG) Solvers in the *Basic Analysis Guide*.

10.2.9. Change in Preconditioned Conjugate Gradient (PCG) Lanczos Eigensolver

The PCG Lanczos eigensolver now uses a single stopping criterion for its internal conjugate gradient (CG) solvers. This criterion is based on the exact norm of the error and is expected to improve result accuracy. Some test cases might show a slight increase in the number of iterations.

In previous releases, internal CG solvers used multiple stopping criteria and stopped iterating when any one was met. Some of these criteria relied on approximate error norms.

for more information, see Conjugate Gradient (CG) Solvers in the *Theory Reference*.

10.2.10. Output behavior of mapdl.run in a *PYTHON block

Within a ***PYTHON** block, issuing `mapdl.run(...)`, no longer displays output by default. Instead, it returns a string that can be printed by issuing `print(mapdl.run(...))`. To restore the pre-

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

vious behavior of automatically printing all output, set the environment variable MAPDL_PYTHON_WRITE_OUTPUT.

10.2.11. Change in Impedance Sheet Formulation for Acoustic Analysis

The updated impedance sheet formulation provides physically consistent behavior for low, matched, and high impedance values. When the sheet impedance is much lower than the characteristic impedance of the fluid ($Z_{sheet} \ll Z_0$), the surface behaves as a soft or limp sheet, allowing pressure release. When the sheet impedance equals the characteristic impedance of the fluid ($Z_{sheet} = Z_0$), the sheet behaves as an almost perfectly transparent interface, producing negligible reflection and allowing the acoustic field to propagate as if no internal surface exists. When the sheet impedance is much larger than the fluid characteristic impedance ($Z_{sheet} \gg Z_0$), the sheet behaves as a rigid interface, strongly restricting normal velocity and introducing significant reflection. For details, see Impedance Sheet Approximation in the *Theory Reference* and Impedance Sheet in the *Acoustic Analysis Guide*.

10.2.12. Change in Default for Ansys Product Improvement Program (APIP)

The Ansys Product Improvement Program (APIP) now automatically enrolls participants. In previous releases, enrollment was off by default and required manual opt-in. You can opt out at any time by following the steps described here: The Ansys Product Improvement Program in the *Operations Guide*.

10.2.13. Change in Behavior Due to Backstress Scaling in the Chaboche Model for Non-Isothermal Loading

For non-isothermal loading, the backstress is now scaled differently. It uses only material parameter C in the nonlinear kinematic hardening model, resulting in changes to stress and strain behavior. In previous releases, non-physical results were observed, such as negative values for plastic work per volume.

10.3. Known Incompatibilities

The following incompatibilities with prior releases are known to exist at Release 2026 R1:

- The force-distributed surface constraint cannot be applied to a single surface node. Instead, use a rigid surface constraint.
- For the exponential pressure-penetration relationship (KEYOPT(6) = 3 on contact elements), calculation of the default value of real constant PZER has been revised to make it more robust and accurate.

10.4. Known Issues

The following issues are known to exist at Release 2026 R1:

- **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

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

stiffness matrix, when used in a Linear Perturbation downstream analysis, may produce reaction forces not satisfying the equilibrium with the external forces.

- **Importing ACIS and CATIA Files** – Check models carefully after importing from ACIS and CATIA as the third party libraries supporting these commands have changed.
- **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
- **On Windows, CRTL+C May Not Terminate All Related Child Processes** – Using **CRTL+C** to terminate a distributed-memory parallel processing (`-dis`) run on Windows may leave child processes running.

  Workaround: To guarantee a complete shutdown, use the **/EXIT** command. If you must use **CRTL+C**, you may need to manually remove remaining child processes using Windows task manager.
- **Downstream Analysis Using Frequency Domain Decomposition May Crash After Hybrid Parallel Use** – A downstream analysis that uses frequency domain decomposition may crash if the upstream analysis was run with hybrid parallel enabled. For example, if the upstream analysis runs with `-np 4 -nt 2` and the downstream analysis runs with `-np 8`, the downstream analysis may crash when the hybrid parallel configuration is restored.

  Workaround: To avoid the downstream crash, use one of the following methods:

  – **Match the upstream parallel configuration**: Launch the downstream analysis with the same number of processes (`-np`) and threads (`-nt`) used in the upstream analysis. Use this method when the upstream analysis explicitly used hybrid parallel.

  – **Disable hybrid parallel**: Launch the downstream analysis with `-nt 1` to disable hybrid parallel. Use this method when the upstream analysis omitted -nt and automatic hybrid parallel was applied.
- **Possible Stress Calculation Failures in Nonlinear Adaptivity Analyses with TNM and Bergstrom-Boyce (BB) Material Models** – Certain shear-dominant high deformation scenarios with the TNM or BB material models in a nonlinear adaptivity analysis can cause stress calculation failures after remeshing.

For issues discovered following publication of this document, see Mechanical APDL in the *Ansys Known Issues and Limitations*.

10.5. Resolved Issues and Defects

The following notable issues and defects for Mechanical APDL have been resolved at Release 2026 R1:

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

ID	Resolution Description
1167416	The automatic hybrid parallel feature no longer oversubscribes hardware on large clusters starting in Service Pack 2025 R2 SP02. This resolves the performance decline observed in the previous release. For more information, see Changes in Automatic Hybrid Parallel (p. 72).
	Automatic initialization in curve fitting for hyperelastic models with Prony series viscoelasticity no longer requires prior installation of the latest Anysys Material Lab (AML) Python module.
1188443	The issue that caused crashes on SUSE Linux Enterprise Server 15 machines has been resolved.
1312569	The performance decline that occurred when running AMD processors on Linux in Release 2025 R2 has been fixed.
1370069	The issue that caused crashes in DMP mode with Intel MPI and NVIDIA GPUs on Linux when running more than 50 number of processes has been fixed.

10.6. Features Removed or Archived

If a legacy Mechanical APDL feature has been removed at Release 2026 R1, 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.

10.6.1. Undocumented Real Constant and Keyoption for Contact Elements

For the contact elements, CONTA172, CONTA174, CONTA175, CONTA177, the following has been undocumented:

- The real constant ICONT (initial contact closure)
- KEYOPT(5)=4 (auto ICONT)

Instead, use the real constant CNOF and **CNCHECK**, ADJUST.

10.6.2. Connection Commands

The following connection commands used to import early versions of CATIA (V4 and earlier) or ACIS files into Mechanical APDL have been archived and are considered obsolete. Instead, use one of the other products like the Mechanical application or Discovery or use one of the CAD/CAE packages supported in Mechanical APDL (see Introduction to the Connection Functionality in the *Connection User's Guide*).

~CATIAIN - Transfers a CATIA v4 or earlier model into the Mechanical APDL program.

~SATIN - Transfers a `.sat` file into the Mechanical APDL program.

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

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

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 Ansys Release 2026 R1:

- **Compiler Update for User-Programmable Features on Linux Systems**. The supported GNU GCC compiler for user-programmable features has been updated to version 12.3.0. For more information, see Compiler Requirements for Linux Systems in the *Ansys, Inc. Installation Guides* and Compilers Required to Work with User Subroutines.

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

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

Aqwa Release Notes

This release of the Aqwa related products contains all capabilities from previous releases. The following enhancements are available in release 2026 R1. 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:

- **Drum Winch for Nonlinear Polynomial Cables**. You can define a drum winch on a nonlinear polynomial cable using the DW10, DWAL, LNDW or NLDW data records.
- **Increased Limit for Number of Elements**. The 64-bit version of Aqwa can accept up to 60,000 elements, of which 30,000 can be diffracting.
- **Maneuvering Force/Moment for Aqwa-Drift**. You can now use the MNVR run-time option for all time response analysis types to request calculation of low frequency maneuvering loads.
- **Mean Drift Force Coefficient Correction**. The mean wave drift coefficient now includes current interaction and viscous effects in accordance with DNV guidance (DNVGL-OS-E301, Position mooring).
- **Radiation Force with Hot Start**. When running a hot start analysis, the motion response at the time steps prior to the hot start time are used in the convolution integral for the radiation force calculation.
- **Ship Position/Course Control**. You can define horizontal motion PID controllers in an Aqwa solver time domain analysis. This can be used to retain the travelling course or maintain a horizontal position of a structure.
- **Warped Elements Check**. The solver checks that nodes of a diffraction quadrilateral panel are in a flat plane.

2. Aqwa Graphical Supervisor (AGS)

There are no new features in the Aqwa Graphical Supervisor.

3. Hydrodynamic Analysis Systems

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

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

- **Automatic Creation of Internal Lid Elements**. Internal lid elements are automatically generated for the analysis by default. This is controlled by the **Generate Internal Lid** option.
- **Display of Disc Normal Vector**. When a Disc element is defined, the unit normal vector is shown in the Details panel.
- **Drum Winch for Nonlinear Polynomial Cables**. You can define a Cable Winch on a nonlinear polynomial cable.
- **Dynamic Control Object**. You can use the new Dynamic Control object to control the motion of a structure, allowing it to move along a specified course or toward a given position or heading.
- **Failure Time of Intermediate Buoy or Clump Weight**. In the Connection Failure object, you can set the **Failure Time** of an intermediate buoy or clump weight in a Time Domain Hydrodynamic Response analysis by selecting the required Catenary Joint for the **Failing Connection** property.
- **Froude Number Warning for Forward Speed**. When a Forward Speed is defined in the Hydrodynamic Diffraction analysis, a warning message occurs when the estimated Froude number exceeds 0.3.
- **Increased Limit for Number of Elements**. The 64-bit version of Aqwa can now accept up to 60,000 elements, of which 30,000 can be diffracting.
- **Maneuvering Force/Moment for Time Domain Hydrodynamic Response Including Drift**. You can now use the Include Maneuvering Force analysis setting for all time response analysis types to request calculation of low frequency maneuvering loads. See the MNVR solver input.
- **Multiple MCOL File Generation**. The **MCOL Hydrodynamic Database** analysis setting allows you to generate multiple hydrodynamic databases based on varying structure positions and orientations by using the template run script file, `WorkbenchScript.py`, included in the standard Windows installation of Aqwa.
- **Parametric studies with pressure mapping**. Ansys optiSLang provides a powerful simulation workflow environment for parameter sensitivity analysis, optimization, and robustness evaluation. For parametric studies involving pressure mapping from a Hydrodynamic system to a Static Structural analysis, optiSLang can perform some additional background steps that are required in the design point update process. optiSLang also delivers more detailed post-processing in parametric response and sensitivity.
- **Pretension Value Definition for Fenders and Cables**. A pretension value can be defined for fenders and all cable types. When the cable pretension value is set directly, the related **Unstreched Length** or **Length** value automatically updates. The cable pretension property is **Initial Tension** for Linear Elastic, Nonlinear Polynomial, and Nonlinear Steel Wire cables, and **Cable Pretension** for Nonlinear Catenary cables. For fenders, an Initial Elastic Force can be defined. Similarly, if this value is set directly, the fender **Size** value automatically updates.
- **Structure Center of Gravity for Starting Conditions**. The Hydrodynamic Response Starting Conditions object now allows the Reference Point to be set to **Structure Center of Gravity**.
- **Upgrade to OpenFAST 4.0.0**. The version of AeroDyn and InflowWind included in the Ansys installation has been upgraded from OpenFAST version 3.0.0 to 4.0.0. For more information, refer to OpenFAST User Documentation, OpenFAST release notes and API changes between versions.

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

It is important to consider the configuration differences when preparing AeroDyn input files and InflowWind input files for 4.0.0.

- **Watch Zone Improvement**. The Aqwa editor, where the defined Watch Zones are rendered, and the window that displays the **Binding Selection** and **Statistic Results** are now synchronized.

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

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

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 2026 R1.

1. New Features and Enhancements
2. Known Limitations

1. New Features and Enhancements

The ACP application includes new features and enhancements at Release 2026 R1 in the following areas:

1.1. Interface to the Ansys LS-DYNA™ Application
1.2. Solid Composite Model with Reinforcing Elements
1.3. Performance Improvements
1.4. Split Modeling Ply into Production Materials
1.5. End of Support for .acp Serialization Format
1.6. Sampling Point for Solid Models
1.7. Improved Search Widget
1.8. Ansys Composite Cure Simulation (ACCS) Updates
1.9. Two Examples of Solid Modeling with Reinforcing Elements

1.1. Interface to the Ansys LS-DYNA™ Application

ACP now includes a refactored LS-DYNA interface based on the Envyo application. This update provides:

- **Expanded Functionality**: Support for a wider range of material, part, and element keywords, as well as additional modeling approaches.
- **Improved Workflows**: Export shell and solid models for further processing in the LS-PrePost application.
- **Direct Model Conversion**: Convert ACP models to LS-DYNA input files using the Envyo application. For more information, see Workflows in the *Envyo User's Guide*.

You can use this interface when ACP runs within the Ansys Workbench™ software or as a standalone application. For details, see LS-DYNA in the *ACP User's Guide*.

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

1.2. Solid Composite Model with Reinforcing Elements

You can now use reinforcing element technology to model layered composite solid models in the Workbench software. This capability supports passing the configured solid model and reinforcing plies from ACP to the Ansys Mechanical™ application for structural or thermal analysis.

The enhancement also enables modeling of layered composites with tetrahedral solid meshes. Previously, solids required layered brick (hex) elements. For more information, see Layup Mapping Objects in the *ACP User's Guide*.

1.3. Performance Improvements

ACP includes several performance improvements, especially for large models:

- Model loading is about 4x faster than previous releases.
- Model updates are about 2x more efficient.
- Solid model processing is about 20% faster, and refreshing a model is more efficient.

1.4. Split Modeling Ply into Production Materials

You can now split a modeling ply with a sublaminate into its production materials. This enhancement simplifies layup refinement workflows, which often start with the same stack of plies for the entire structure and then refine it step by step. The initial step can be accomplished using sublaminates, but previously, inserting local patches required manually splitting those sublaminates into separate plies.

1.5. End of Support for `.acp` Serialization Format

The legacy `.acp` serialization format is no longer supported. The `.acph5` format, introduced in Release 2019 R1 (19.3), replaces `.acp` and is required for compatibility with current and future releases.

How to upgrade older projects

To upgrade projects created with ACP 19.2 or earlier:

- Use any version of Workbench software between 2019 R1 and 2025 R2.
- Open and close all ACP setup and results components, then save the project.

Removed functionality

Export and import of composite definitions to `.acp` files is removed. For individual ACP objects, use the `serialize()` method to obtain Python code for creating the object. To duplicate multiple interdependent objects or copy them between models, use the PyACP `recursive_copy()` function.

For details, see linkShare and Copy Composite Definitions Between Models in the *ACP User's Guide*.

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

1.6. Sampling Point for Solid Models

The Composite Sampling Point tool in the Mechanical application can now scope to an ACP solid model. This tool provides access to results from the bottom to the top of the laminate. Visualizing through-the-thickness distribution of stacking sequence, strains, stresses, and failures helps you understand laminate behavior and optimize designs.

For more information, see Composite Sampling Point Tool in the *Mechanical User's Guide*.

1.7. Improved Search Widget

The ACP object tree search widget now supports seamless object selection from a filtered tree. This improvement makes the UI faster and simplifies handling large models with many named selections.

1.8. Ansys Composite Cure Simulation (ACCS) Updates

Updates include:

- Improved support for multiple curing cycles and steps, including better import of the initial degree of cure into ACCS.
- Added support for porosity in the resin transfer molding (RTM) solver.
- Improved inlet definition in RTM workflows.
- Added a **Post Fill Phase** for the RTM solver.

For more information, see Ansys Composite Cure Simulation Documentation.

1.9. Two Examples of Solid Modeling with Reinforcing Elements

The following two examples are now available:

- **T-Joint Example:** Demonstrates the standard workflow for building a layered 3D composite structure using reinforcing elements.
- **Fan-Blade Example:** Demonstrates how reinforcing elements integrate with Imported Plies that originate from CAD surfaces rather than reference surfaces.

2. Known Limitations

General ACP application and release-specific limitations can be found in Known Limitations in the *ACP User's Guide*.

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

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

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 2026 R1 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*.

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

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

Additive Manufacturing

Additive Suite products include new features and enhancements at Release 2026 R1 in the following areas:

1. PBF Process Simulations
2. DED Process Simulations
3. Sintering Process Simulations
4. Distortion Compensation
5. Changes in Product Behavior
6. AM-Related Mechanical APDL Commands
7. PyAdditive

1. PBF Process Simulations

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

- **Calibration improvements**:
  - **Quick Calibration – Cutoff height options**: The Quick Calibration feature now offers more flexibility when specifying support cutoff height for the cantilever geometry. Choose a cutoff height between 0 - 8 mm (in 1 mm increments) from the base plate. Previously, the cutoff height was predefined and fixed at 3 to 4 mm.
  - **Quick Calibration – Import CSV file**: Save time on setup! You can now directly import a CSV file containing location-specific distortion measurements. This means you can process your setup data in a spreadsheet beforehand for easy import into the calibration wizard.
  - **Full Simulation Calibration – Anisotropic calibration**: Anisotropic calibrations are now supported when performing full simulation calibrations.
- **Electron Beam Melting**: You can now simulate an Electron Beam Melting (EBM) process, also known as Electron Beam Powder Bed Fusion (E-PBF). While both E-PBF and L-PBF involve fusing powdered material layer by layer to build parts, they differ significantly in their mechanisms and applications. To accommodate an EBM simulation, the following enhancements have been added to the PBF simulation capability in the Mechanical application:
  - *Build sequence*: New **Build Sequence options** allow you to define distinct heating steps in an EBM process, for different preheats, for example. You can also define location specific process parameters, allowing different portions of the part to have different process parameters for the same layer. The heating method can be temperature or power. Build sequence options are controlled in Build Settings.

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

  - *Build environment*: You can now account for heat losses from **radiation**, which is the dominant heat transfer mechanism in a vacuum. With radiation effects enabled, a surface boundary condition is applied to the top of a newly laid layer (build step) and the top of the build (cooldown step). Specify an Emissivity value between **0** (no radiation) and **1** (perfect blackbody emitter) to define the surface's ability to emit radiation. Radiation can be specified in the PBF Setup Wizard or in Build Settings.

  - *Multiple materials*: Simulations are no longer restricted to using a single material for the build (part and supports). You can now model PBF simulations with multiple materials. For a Thermal-Structural AM simulation, the melting temperature for each part geometry must be defined in Engineering Data. A **Representative Material** option will be displayed in AM Process when multiple material assignments on the part geometries are detected. Choose the material that is most representative of the model as a whole, specifically for thermal properties.

    → Predefined Support structures can now be assigned a different material than the part geometry they are supporting.

    → For STL and Generated Supports, the material will be the same as the part they are scoped to.

  - *Modeling of powder (Beta)*: For voxel meshes, you can generate powder padding around the part and extend the powder below the part to the base plate.

  The updates to the underlying **AMBUILD** Mechanical APDL command made at the last release enable these EBM enhancements in the Mechanical application at this release.

- **Multi-step heat treatment**: The PBF Setup Wizard now supports complex, multi-stage thermal cycles, moving beyond the standard single ramp-and-hold sequence. This allows for high-fidelity simulation of sophisticated industrial heat treatment schedules.

  - **Advanced Cycle Configuration:** A new **Multiple (Use Table)** option enables the definition via a table of intricate cycles with various heating ramps, soak times, and cooling rates.

  - **Granular Solver Control:** New **Time Stepping** controls allow for manual substep adjustments to ensure accuracy during critical thermal transitions.

  - **Automated Workflow:** The wizard intelligently detects and preloads existing convection loads into the table, seamlessly transitioning between simple and multi-step configurations.

- **HPC Platform Services (HPS) for AM simulations**: You can now leverage Ansys HPC Platform Services (HPS) to accelerate Additive Manufacturing simulations. Inherent Strain and Thermal-Structural simulations with heat treatment steps can be solved on HPC Platform Services. This feature requires that you set up a Solve Process Queue for the required HPC server details.

- **UI changes**: You'll notice in the UI that items are renamed to be PBF rather than LPBF as a subtle reminder that *electron* beam simulations are now possible, and not just *laser* beam simulations. For example, the LPBF Setup Wizard is now the PBF Setup Wizard, the LPBF Hotspot result item is now the PBF Hotspot, and so on.

  Additionally, the AM-specific result items are now consolidated under two new buttons in the PBF Process ribbon, as follows:

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

Thermal PBF Results

Hotspot

Hotspot Time Correction (Beta)

Heat Flux (Beta)

Structural PBF Results

Recoater Interference

High Strain

Shrinkline (Beta)

- **Visualize cutoff direction**: An arrow has been added in the graphics window to indicate the direction of base/support cutoff when directional cutoff is chosen (Analysis Settings, Base Removal Type = Directional).
- **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 (Beta) 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.
- **Visualize conformal mesh (Beta)**: A new visualization tool is now available for viewing AM simulation results on a conformal body instead of voxel mesh elements. After completing the AM process simulation, select the desired result object under Solution and click Evaluate. Once evaluation is complete, click the **Visualize Conformal Mesh (Beta)** button on the PBF Process ribbon.
- **PBF Heat Flux result (Beta)**: For AM thermal-structural simulations, a new **Heat Flux (Beta)** result item allows you to evaluate the maximum heat flux for each element at the conclusion of every layer's deposition step. You can toggle between Total Heat Flux and Directional Heat Flux. Choose between contour plots for gradient visualization or vector plots for flow direction. Visualize data from either the first cooldown step or the last cooldown step.

  The Heat Flux (Beta) result can be added to a Thermal AM Analysis System and requires the "Calculate Thermal Flux" output control to be on. If adding the result before the solve, this will be done automatically.
- **PBF 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.

2. DED Process Simulations

Enhancements for directed energy deposition (DED) simulations include the following:

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

Additive DED Add-in in the Discovery™ Application

Introduced at the last release, Discovery's Additive DED add-in allows you to generate geometry models directly from a tool path file, such as G-code. This Path to Part method offers more accurate views of print characteristics, like deposition volume and geometric feature details, compared to using nominal CAD designs. More functionality has been added to the add-in at this release:

- **Import CLS file format (Beta)**: You can now import a file in a Cutter Location Source (CLS) format. Specify the Paint Index used for deposition.
- **Multiple machine types supported**: You can now specify either a Three-axis or Multi-axis machine in the Path tool. Inputs for Multi-axis machine include X-axis Rotation, Y-axis Rotation, Z-axis Rotation, Rotation Sequence, and Tool Direction.
- **Export material orientation from tool path file**: You can now easily export material orientation data directly from the G-code/CLS file into the Mechanical application. This new function exports the Euler angles at every path point into a `.CSV` file. In the Mechanical application, this data is used to define local coordinate systems, which are necessary for applying realistic anisotropic (direction-dependent) material properties to your simulated part. The Interpolation Distance input is available to automatically generate interpolated points, achieving a finer resolution of local coordinate systems than the defined path points provide.
- **Visualization of tool path orientation**: A new object called Tool Path is added to the tree upon import of your G-code or CLS file (Beta). This object stores the complete tool path and the specific orientation of the printer head at every position. Beyond viewing path curves, you can now visualize the deposition direction at any point—a critical feature for accurately interpreting how the bead profile is applied throughout the build.
- **New bead profiles added**: When using the Path to Part tool, in addition to the standard Circular bead cross section, you can now specify a Rectangular or an Elliptical bead profile.
- **Scripting supported**: You can now use Discovery's Script Editor to automate the workflow steps when using the Additive DED add-in. Record the steps in the workflow, adjust parameters as desired in the Script Editor, and replay the script any time. This technique allows you to customize the bead profile beyond the provided profiles (Circular, Rectangular, or Elliptical). Simply change the coordinates in the `profile` line of code in the script.

DED Process Add-on in the Mechanical Application

- **Multi-material deposition**: You can now simulate deposition of multiple materials by simply defining distinct bodies and assigning the desired materials.
- **Hotspot result item**: A new Hotspot result item has been added for calculating residual hotspots in the DED process. The hotspot shows the maximum temperature in each depositioned cluster of elements immediately before the next cluster is activated.

3. Sintering Process Simulations

Enhancements for sintering simulations using the Sintering Process Add-on include the following:

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

- **Location-dependent green density improvements**: We've added the ability to specify variable density in the Sinter Material object using tabular entry, simplifying the process of importing experimental green density data for sintering simulations.

  This feature is available in the Sintering Process Add-On by choosing Sinter Material > Initial State Data = Tabular under Definition.

- **Alumina material model**: A new Alumina material model is now available, allowing you to model one of the most versatile and widely used technical ceramic materials due to its exceptional combination of mechanical, thermal, and electrical properties. The material model uses the newly added Ashby modifier for sinter stress and the newly added Pore Pinning modifier for grain-growth kinetics with 4^{th} power for grain size.

- **Advanced grain-growth model**: A new general power law model for grain-growth kinetics is now available, enabling more accurate simulation of normal grain growth during sintering. Unlike the previous parabolic model, this approach supports variable growth exponents, making it suitable for diverse materials such as ceramics and metals used in powder metallurgy.

  The feature is available by choosing Sinter Material > Model = Advanced under Grain-Growth Kinetics.

- **New sintering stress evolution models**: Three new stress evolution models are available for accurate modeling of sintering stress. This enhancement builds upon existing stress calculation by adjusting the calculated stress. The new models are based on work by McMeeking & Kuhn, Ashby, and Shinagawa.

  Choose an option on Sinter Material > Stress Modifier under Sintering Stress.

4. Distortion Compensation

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

- **Option to combine STL files when exporting compensated STL files**: A new option, **Combine Bodies** (Yes/No) merges the exported distortion-compensated STL files into a single file. This is particularly useful for DED simulations containing multiple cluster bodies. When a Single Compensation object is scoped to all clusters, it uses the refaceting settings to generate a single, unified STL file consisting of all merged clusters.

5. Changes in Product Behavior

Release 2026 R1 of the Mechanical application includes AM-related enhancements that cause product behavior to differ from earlier releases, as explained below:

Powder Bed Fusion (PBF)

- The previous limitation of a single build material has been removed. PBF simulations now support multiple materials defined via different bodies.

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

Directed Energy Deposition (DED)

- In this release of the Additive DED Add-in in the Discovery™ Application, the **G-code** tool under **Import** has been renamed to **Path** to reflect support for an additional import format: CLS (Beta).
- In this release of the Mechanical application, a boundary condition check has been added for add-ons during the solve step for each system/environment.

  To pass this check, DED simulations now require a "DED_Loads" placeholder command snippet in both the Transient Thermal and Static Structural systems. These command snippets are automatically inserted by the AM Process for DED object, including when reopening a previously solved archived project from an earlier release.

  Note: After importing an archived project, the simulation state may display a yellow lightning bolt, indicating it is in an obsolete state. However, the results are still valid and can be retrieved.
- The previous limitation of a single deposition material has been removed. DED simulations now support multiple materials defined via different bodies.
- The requirement to use a Cartesian mesh for multi-axis simulations has been removed. You can now utilize a wider range of element types—including Hexahedral (Hex) and Tetrahedral (Tet)—to better represent complex geometries and organic shapes in multi-axis DED workflows.
- The `DEDClusterOutput.json` file generated during DED clustering has been updated to start with an index of 1. This replaces the previous 0-indexed format to provide better consistency across the simulation workflow.

Sintering

- With the addition of the ability to specify variable density in the Sinter Material object using tabular entry:
  - Node-based **INISTATE** mapping is used. Subsequent mappings are limited to node-based algorithm.
  - The UF01 field variable is reserved by the Sinter Material object for specifying mass density.

6. AM-Related Mechanical APDL Commands

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

- **AMENV** – Specifies the build-environment thermal boundary conditions in an additive manufacturing (AM) analysis. Use the new *RADOPT*, *EMIS*, and *SBC* options to define radiation boundary conditions in a PBF simulation. Radiation is important when simulating an EBM process. When radiation is enabled, heat losses from radiation are applied to the top of a newly laid layer during the build and cooldown process steps.

7. PyAdditive

PyAdditive is a Python client library for the Ansys Additive server. The Ansys Additive server is distributed with the Additive option of the Structures package in the Ansys unified installation.

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

Use PyAdditive to run single bead, porosity, 2D microstructure, 3D microstructure (Beta), and thermal history simulations, as well as custom material tuning. An Ansys Additive Suite license is required to run the simulations and tune materials. End-to-end examples show how you can use PyAdditve. You can download these examples as Python files or Jupyter notebooks and run them locally. For more information, see the PyAdditive documentation.

New in PyAdditive

Newly released functionality for PyAdditive that is included in the 2026 R1 Ansys Additive server include:

- **Ring mode (Beta)**: A new *ring mode* (ring-shaped) beam profile option has been added to the laser modeling capabilities. This feature uses a mathematical representation to simulate how various donut-type laser intensity distributions influence melt-pool behavior in additive manufacturing.

  The ring mode includes **seven indices (0–6)** that adjust the relative intensities of the primary and secondary laser peaks. Index 0 closely aligns with the traditional Gaussian distribution. As the index increases, the primary peak intensity decreases while the secondary peak becomes more dominant, resulting in progressively stronger ring-shaped profiles.

  Ring mode beam shapes are commonly used to generate wider melt pools, increasing productivity during bulk build regions. They can also help mitigate keyhole porosity in certain materials, such as AlSi10Mg.

- **Laser defocusing heat source model (Beta)**: We've introduced a new Laser Defocusing Heat Source Model (Beta) that allows you to simulate how heat is distributed when a laser beam is intentionally offset from its focal point. This model is useful for processes such as welding, cutting, additive manufacturing, and surface treatments where broader, lower-intensity heat input is desired.

  In this initial release, the model supports positive defocus only, meaning the laser's focal point is positioned above the material surface. This creates a wider, shallower laser spot, enabling you to evaluate conditions where reduced energy density and gentler heating are important.

  Defocusing affects energy density, melt pool geometry, thermal gradients, and cooling rates.

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

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

Sherlock

Introduction

This document provides a summary of all new features, product enhancements, limitations, and bug fixes in the 2026 R1 release of the Ansys Sherlock™ electronics reliability prediction software.

1. Known Issues
2. New Features
3. Resolved Issues and Limitations
4. Scheduled Updates
5. General Enhancements
6. API Updates
7. Bug Fixes
8. Third Party Dependency Changes

1. Known Issues

The Sherlock application may not be compatible with certain features of Abaqus version 2025 and later. (1407200)

2. New Features

You can now import and convert thermal signal data into a Thermal Event of a given Life Phase. The Life Cycle Viewer and the Report feature have been updated to include charts and tables which visualize the imported thermal signal data. See Import Thermal Signal (Rainflow Cycle Counting) in Sherlock User's Guide. Currently, the 4D chart feature is not supported on Linux

3. Resolved Issues and Limitations

For Sherlock release 2026 R1, the following issue has been resolved: When exporting a blocked CDB model from the Sherlock application with solder balls modeled as shells and beams, the solder balls were missing from the exported CDB file. The CDB export feature now works correctly.

4. Scheduled Updates

The DFMEA feature will be sunset in the next release, 2027 R1, of the Sherlock application.

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

5. General Enhancements

Links:

5.1. Scoring Updates
5.2. Updates to Sherlock Component Models
5.3. Updates to Sherlock Analysis Modules
5.4. Solder Updates
5.5. Updates to Ansys Workbench/Mechanical Application Integration
5.6. GUI Updates
5.7. Performance Improvements
5.8. Other Enhancements

5.1. Scoring Updates

The color-coded scoring system has changed for the following analysis modules: Solder Joint Fatigue, PTH Fatigue, Semiconductor Wearout, Harmonic Vibe, and Random Vibe:

- The numeric score has been removed from the analysis results.
- The color-coding system used in the results is now based on the probability of failure ratio which you can configure in **Main Menu >Settings > Score**.

> **Caution:**
>
> Because the color-coding system used in the results is now based on probability of failure, the color applied to components may differ from earlier versions of the Sherlock application.

- The overall numeric score has been removed from the Results Summary and reports. The numeric score per analysis module has also been removed from reports.

See the chapter Color-Coded Scoring System in the Sherlock User's Guide for a complete explanation of the new scoring system.

5.2. Updates to Sherlock Component Models

Leaded Model:

- The Lead Height property no longer applies to parts with stub leads.
- A new package property called Package Compliance Factor is now used in the combined stiffness calculation for all leads. See Package Compliance Factor in the Sherlock User's Guide. To enable editing the **Package Compliance Factor** field in the **Part Editor**, go to **Settings > Advanced > Part List > Package Compliance Factor** in the Sherlock application.
- The combined stiffness calculation has been updated for lead layouts that are quad, rectangular, or circular.

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

- A shear spring model is now used for calculating the stiffness of stub leads.
- In the Package Data Base, the data for stub lead packages has been updated for the following types: SON, USON, SOD, OTHER, SOT, TO, and SM.

BTC/IMS Packages:

- The Sherlock application now detects the IMS routing pattern from top and bottom copper layers for BTC packages. This is used to guess the Design Factor part property. Detection happens automatically when component layers are updated or a top or bottom copper layer is processed. See BTC Packages in the Sherlock User's Guide.
- The detection of BTC package type based on the ECAD package name has been enhanced.

BGA and CGAs

- The BGA and CGA solder models have been updated: The reduced shear modulus is adjusted to account for the distance to neutral point (DNP). Also, the corner overmold material is adjusted to account for the DNP.
- The detailed BGA package model option is no longer beta. It is now a standard feature and can be enabled in **Settings > Meshing**. See Detailed Package Modeling (Thermal Mech Life Prediction for BGAs) in Sherlock User's Guide.

CC and LCCCs

- Previously, the chip component (CC) and leadless chip carrier component (LCCC) models were producing identical results because they used the same critical length formulation. To improve model accuracy and better reflect physical behavior, the Sherlock application now uses a revised shear stress formulation and an updated definition of critical length.

The Sherlock Part Library

- The Sherlock 2026 R1 Part Library update contains 2,236,857 parts.

5.3. Updates to Sherlock Analysis Modules

- The option to use contact elements in place of constraint equations for FEA models is no longer beta. It is now a now a standard feature which you can enable in **Settings > FEA**.
- You can now calculate Time-to-Fail results at any PoF (probability of failure) for the following analyses: PTH, Component Failure Mechanism, Harmonic Vibe, Random Vibe, Mechanical Shock, ICT Analysis, Solder Fatigue, and Thermal Mech. The probability percentage value can be adjusted in the **Advanced Settings** dialog. In the Sherlock User's Guide, see the entry Life Metric At (%) under **Advanced Settings > General**.
- An improved Stackup Calculator has been added. For new and imported stackups, you can enable this feature in the **Advanced Settings** dialog. When enabled, new calculations are used for the CTE_{xy} and the E_{xy} for conductive layers and the CTE_{xy} for the stackup.
- Solder Fatigue analyses for BGAs and CGAs are now executed successfully even if a part's Ball Modeling property is set to NONE.

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

5.4. Solder Updates

- The secant modulus for eutectic SnPb solder has been recalibrated using Anand's viscoplastic model, replacing the previous material representation to improve accuracy under thermomechanical loading.
- Solder materials SAC 302 and SenjuM758 have been added.

5.5. Updates to Ansys Workbench/Mechanical Application Integration

- A new Sherlock native extension—an app accessible from within the Ansys Mechanical application—integrates some of the Sherlock pre-processing capabilities directly into Mechanical. This enables you to perform select Sherlock functions without leaving the Mechanical environment. See the Sherlock User's Guide, The Sherlock Native App in Ansys Mechanical.

  The following enhancements were added to support the new app:

  – The Sherlock application now has the ability to hide the GUI at startup.

  – New APIs (p. 104) have been added to toggle specific parts of the GUI.

- Additional notifications have been added to the gRPC bi-directional streaming between the Workbench and Sherlock applications. Sherlock notifies Workbench when:

  – A Modeling Region is added or deleted.

  – The Edit Properties dialog is used to change any properties of a Mechanical Part, Modeling Region, Potting Region, Cutout, or Mount Point.

  – A Mechanical Part, Modeling Region, Potting Region, Cutout, or Mount Point is moved, rotated, scaled, copied or merged.

  – A new CCA is created from a Modeling Region.

- When importing FEA results from the Ansys Mechanical application, you can now import pre-stressed static structural analysis results with large deflection enabled. See Analysis Properties: Imported Data Source in the Sherlock User's Guide.
- Added a materials XML used when exporting isometric materials of an FEA model or a Trace Reinforcement model to Workbench. Previously, the data was exported by using a Workbench API in the WBJN files.
- You can now export orthotropic PCB properties when exporting WBJN files. See Set PCB Modeling Settings in the Sherlock User's Guide.

5.6. GUI Updates

- The **Thermal Mech Analysis Properties** dialog now cleanly displays a large number of Thermal Events.
- A **Recently Used Files** selector has been added to the **Import Stackup** dialog.

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

- In the **Parts List Validation Properties** window, there are no longer preset selections under **AVL Options**.
- The Sherlock **Project Tree** now supports informational images. When you add an image file to a project, the Sherlock application now associates it as an info image by default.

5.7. Performance Improvements

- Performance has been improved when importing GDSII files.
- The beta option Polygon Merge Performance has been integrated into the base program. This improves performance when polygons are merged during Trace Model Generation.
- The GUI is now more responsive when analysis results are cleared as a result of Life Events being changed.

5.8. Other Enhancements

- Logo images have been updated for reports.
- When importing an IPC-2581 archive, a warning appears if the archive does not implement the FULL or USERDEF function mode as noted in the IPC-2581 Import Tutorial in the Sherlock User's Guide.
- The **Use Part Body Weight** option is no longer an Advanced Setting. It is now a Part Property. To use the part body weight to compute material density, you must enable **Use Part Body Weight** for each part in the Part Editor.
- When laminates not found in the Sherlock Laminate Library are imported into a stackup, the laminates are displayed as errors in the stackup. This serves as a reminder to update the stackup or to add the laminates to the Library.

6. API Updates

Links:

6.1. General API Improvements
6.2. New APIs
6.3. Modified APIs
6.4. Deprecated APIs
6.5. Previously Deprecated APIs Removed

6.1. General API Improvements

When API calls are executed that are analogous to actions performed in the Parts Editor and Layer Viewer, the Workbench application now receives notifications via the gRPC bi-directional stream. The actions and notifications match those that occur when the Sherlock GUI is used.

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

6.2. New APIs

- **getTestPointProperties()**: Returns the properties for a specified set of test points.
- **importCopperFiles()**: Imports copper layer files to a Sherlock project.
- **getICTFixturesProperties()**: Returns the properties for a specific set of ICT Fixture IDs.
- **updateTestPoints()**: Updates the properties of one or more test points in a CCA.
- **importThermalSignal()**: Enables the importing of thermal signal data.
- **updateICTFixtures()**: Updates the properties of one or more test points in a CCA.
- **importPartsToAVL()**: Import any supported file type containing parts to be added to the AVL (Approved Vendor List).
- **saveHarmonicProfile()**: Saves a Harmonic profile to a `.csv` or `.dat` file.
- **saveRandomVibeProfile()**: Saves a Random Vibe profile to a `.csv` or `.dat` file.
- **saveShockPulseProfile()**: Saves a Shock Pulse profile a `.csv` or `.dat` file.
- **saveThermalProfile()**: Saves a Thermal profile a `.csv` or `.dat` file.
- **updateLifeCycle()**: Updates the Life Cycle of a project. It also allows you to save unsaved results files as the GUI does. It overwrites existing results files with the same name.
- **updateLifePhase()**: Makes changes to a specific Life Phase. It also enables you to save unsaved results files if an archive file name is included.
- **updateMountPoints()**: Updates mount point properties.
- **deleteEvent():** Deletes a Life Cycle Event.
- **deletePhase()**: Deletes a Life Cycle Phase.
- **importThermalSignal()**: Updated rainflow cycle binning so it uses separate binning parameters for range, mean, and dwell.

6.3. Modified APIs

- **getStackupProps()**: Updated to allow values to be retrieved based on the use of the improved Stackup model, thanks to the new **improveStackupModelEnabled** parameter
- **exportFEAModel()**: Updated to allow the material elasticity type to be specified when exporting WBJN files.
- Added validation for the side property in the **updateMountPoints()** API. If an invalid side is inputted, the request fails now.
- The **addCCA()** API no longer waits thirty seconds to return. It now returns once the CCA has been successfully added.

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

6.4. Deprecated APIs

- Stackup service:
  - **ListLaminateThicknessUnits()**
- Lifecycle service:
  - **ListFreqUnits()**
  - **ListTempUnits()**
  - **ListAmplUnits()**
  - **ListShockLoadUnits()**

6.5. Previously Deprecated APIs Removed

- Sherlock Common Service:
  - **getSolders()**
- Sherlock Parts Service:
  - **getPartLocation()**

7. Bug Fixes

- Corrected a mesh generation issue where mount hole meshes could sometimes overlap the PCB mesh instead of merging with the PCB mesh when very small mesh sizes were used.
- Exported blocked CDB files using Shells and Beam elements for BGA's now correctly define the beam element type in the CDB file.
- When running an analysis with leaded parts that do not have die properties, the Sherlock console no longer displays a warning about missing die properties.
- Corrected the color display for component footprints in the Layer Viewer so the color of component analysis results appear as expected when overlaid on the components.
- When importing ODB++ files, a parsing error is no longer displayed when user-defined attributes are encountered.
- When importing a stackup, the Sherlock application might not properly associate or clear errors, resulting in future imported stackups failing. This has been corrected.
- In some cases, when importing a stackup or editing a stackup layer, stackup rows appeared to be missing or duplicated until the project was reopened. This has been corrected.
- When a stackup contained errors and the errors were cleared, the stackup would no longer display the error tab if errors were subsequently introduced, such as after an import. This error has been corrected.

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

- User-defined stackup properties are no longer reset when a CSV stackup is imported.
- Corrected an issue where IPC-2581 cutouts were getting converted with the wrong measurement units, causing them to not be rendered improperly in the Layer Viewer.
- When trace models are generated from IPC-2851 archives, line paths are now properly filled.
- Corrected an issue where an imported Random Vibration analysis using nodal lead forces would fail to run to completion when there was a missing PSD file.
- When exporting part material density values using part body weight, part corner shape is no longer taken into account when computing density.
- Corrected an issue with the exporting of trace reinforcement models that could cause the length of vias to be incorrect.
- In the Package Editor tabs, the application no longer allows you to save blank unit fields for **Package**, **Die**, **Pad**, **Ball**, or **Flag**.
- The Solder Editor was not validating CC Calibration Factor values which permitted the field to be blank. This prevented the exported solder from being reimported into the Sherlock application. The blank field also prevented the **getSolderInfo()** API from returning solders. This bug has been fixed.
- When packages were exported, the exported Lead Height was set to 0 for packages that relied on Guessed Lead Height values. This has been corrected.
- Fixed a bug causing illegible text on the Analysis Errors tab for the Japanese font.
- When editing conductor layers in a stackup, you can now change the Conductor Thickness units when multiple layers with different Thickness values are selected.
- When importing a stackup CSV file, the Sherlock application no longer recomputes the layer material property values based on matching Sherlock material and Laminate Library properties.
- Updated the spacing around the Help field dialog to prevent trimming of font glyphs.
- When importing laminates, the Manufacturer, Grade, or Material are no longer forced to upper-case.
- After changing the name of a Laminate, you no longer have to restart the Sherlock application for it to appear in the Stackup Laminate Chooser field in the Stackup Editor.
- Corrected issues with importing ODB++ compressed files disguised as GZip files.
- Resolved a bug which resulted in a series of number formatting exception errors issue when importing an IPC-2581 archive.
- Updated solder Senju M705 CC Calibration.
- Corrected an issue where the order of the connected points in the Material Temperature Calculator was based on the X values (temperature) rather than the order they were calculated. Also, a limit of 300 points was added.

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

- In the Strain Map analysis, the Maximum Strain might not be selected under a component footprint in the following situation: The component strain is one of the image pixels at the maximum X or Y location of the component, and the component did not fully cover the pixel. This has been corrected.
- A formula error for the IMS solder model has been corrected. The strain energy calibration at max temp referenced hot-dwell effective package stiffness instead of cold-dwell effective stiffness.
- The following corrections have been made to the shells and beams CDB export feature:
  - The error "Object doesn't support this property or method" no longer occurs when exporting to the Workbench application.
  - Corrected the problem of contacts not being created for one-element components.
  - When exporting layer models, PCB surface Named Selections are now assigned correctly.
- Previously, the Sherlock application was not correctly parsing Gerber aperture definitions larger than three digits, resulting in copper layers improperly drawn. This issue has been corrected.
- Corrected the tutorial file `Assembly Tutorial.zip` so the imported Parts List contains the correct refDes data values.

8. Third Party Dependency Changes

- JAXB-RI 4.0.5 has been added to Sherlock
- Java OpenGL and Gluegen libraries have been updated to version 2.5.0.

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

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

Release Notes

The 2026 R1 Ansys Forming release contains all of the capabilities from previous releases plus many new features and enhancements. Review changes and new capabilities below:

1. Geometry & Preprocessing
2. Forming Process Enhancements
3. Solver & Performance Improvements
4. Postprocessing & Reporting
5. Documentation

1. Geometry & Preprocessing

New features and enhancements include:

1.1. Sketch Simple Curves Directly in Ansys Forming
1.2. Edit Curves for Faster, More Flexible Design
1.3. Guided Pin Generation for Greater Flexibility in Tool Design
1.4. Transition to Parasolid Geometric Kernel

1.1. Sketch Simple Curves Directly in Ansys Forming

You can now create simple curves within Ansys Forming, reducing dependency on external CAD tools during the design process. Sketch simplifies geometry preparation for forming simulations and supports common curve types such as points, lines, rectangles, circles, and splines.

Sketch allows you to:

- Define blank shapes, drawbeads, and trimming paths directly in the application.
- Apply geometric constraints and dimensions for precise control.
- Create curves in different planes.
- Interact with your sketch as you create by snapping points, dragging curves, and more.

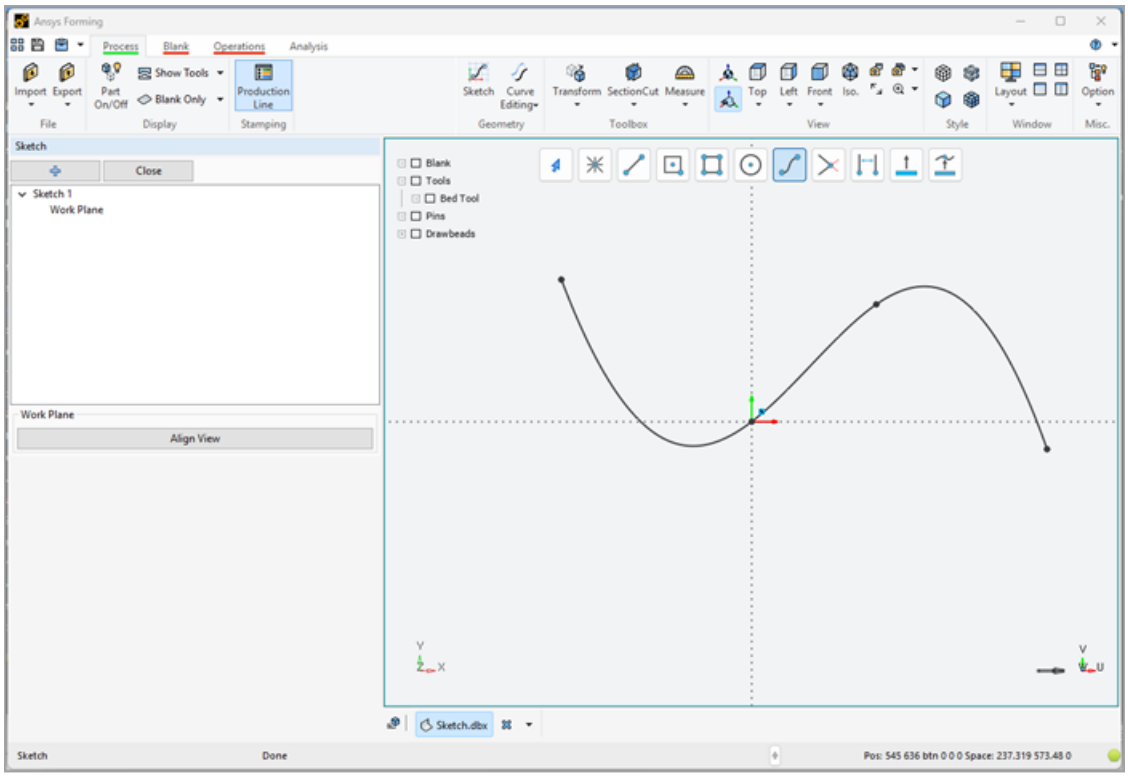
This feature pairs well with another new feature, curve editing (p. 110), enabling quick modifications to imported or sketched curves. Together, these tools provide a complete geometry workflow — import, sketch, and edit — all within Ansys Forming.

Access **Sketch** from the **Geometry** section of the ribbon.

Review Sketch Curves to learn more.

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

Figure 1: Sketch Interface with Sample Spline



1.2. Edit Curves for Faster, More Flexible Design

Curve Editing enables you to modify imported or sketched curves directly in Ansys Forming, reducing the need to return to external CAD tools and accelerating design iterations.

Available editing options include:

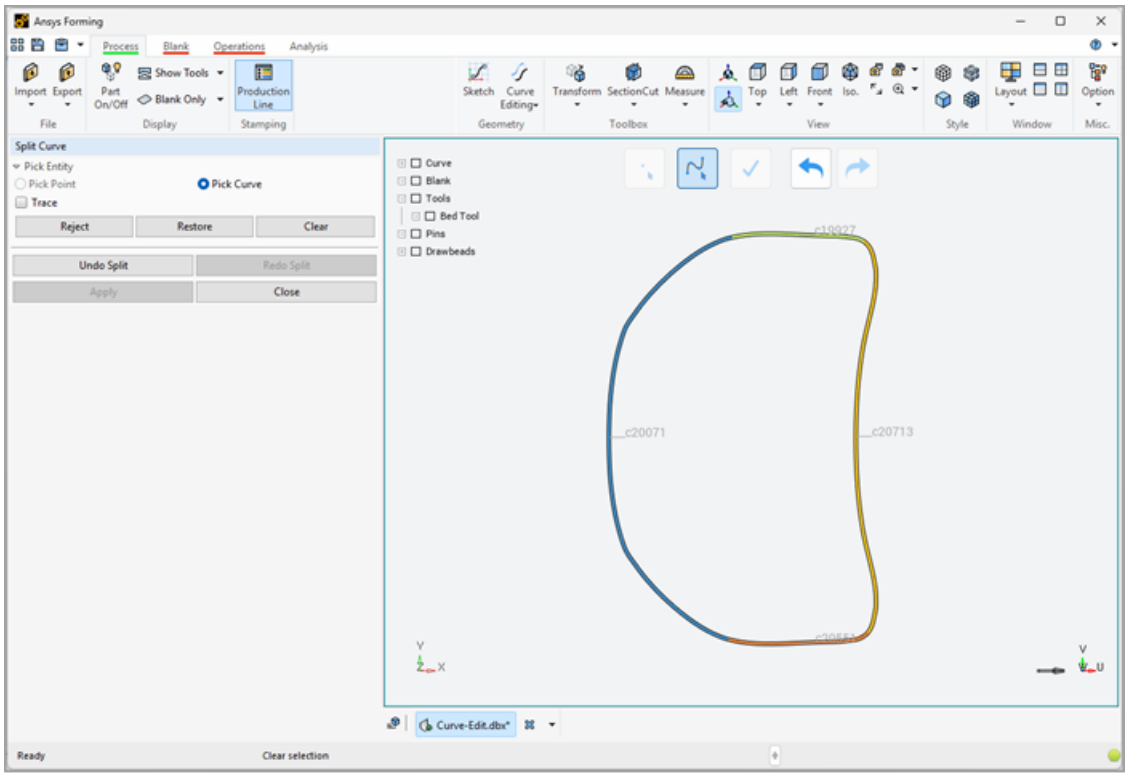
- Split curves into multiple sections.
- Extend curve ends to meet design needs.
- Merge or bridge curves for smooth transitions.
- Locally shift segments for precise adjustments.
- Offset a new curve on either side of an existing 2D.
- Offset a 3D curve over surfaces.
- Extract the boundary of a surface to use as a curve.

Access **Curve Editing** from the **Geometry** section of the ribbon.

Review Edit Curves to learn more.

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

Figure 2: Curve Editing Interface with Sample Curve Split into Four Segments



1.3. Guided Pin Generation for Greater Flexibility in Tool Design

Guided Pin Generation allows you to design guide pins directly within Ansys Forming. You can choose from multiple generation methods — Free Picking Point, Center of Hole Curve, or Boundary Alignment — and adjust parameters such as pin direction, radius, and length.

This feature streamlines setup and improves flexibility for complex tooling requirements. All pin generation options are accessible from the **Guide Pins** task panel, ensuring a smooth workflow.

Review Add Guide Pins to learn more.

1.4. Transition to Parasolid Geometric Kernel

Ansys Forming now uses the Parasolid geometric kernel for geometry handling and operations. Note the following important updates:

Forming Processes Remain Unchanged

All previously defined forming processes remain unchanged. If you open a project you started in Ansys Forming 25R2 or earlier and *only* modify the forming process without redefining tools or blanks, no action is required.

You May Need to Reimport Geometry

If you open a project you started in Ansys Forming 25R2 or earlier and you must change any geometrical data, such as surfaces, curves, and similar entities, **you must reimport your geometries**. These will not be automatically converted to the new kernel.

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

Benefits of the new kernel include:

- Support for native Parasolid files.
- Foundation for advanced features such as Curve Editing and Guided Pin Generation.
- Improved robustness and accuracy for geometry operations.

This change lays the groundwork for future enhancements in geometry workflows.

2. Forming Process Enhancements

New features and enhancements include:

2.1. Improved Support for Solid Elements
2.2. Tipping Support for One Step Projects
2.3. Double Drawbead Now Supported
2.4. Define Multiple Offset Values for Local Tooling Regions
2.5. Flexible Gravity Simulation
2.6. Operation Suspension in Forming Workflows

2.1. Improved Support for Solid Elements

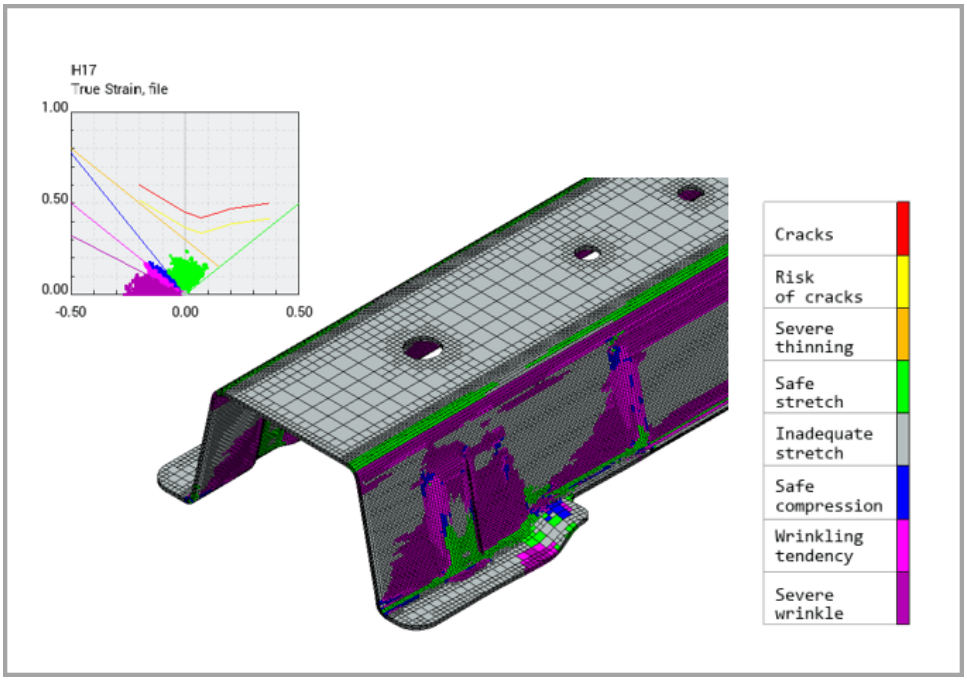
In the 25R2 release, Ansys Forming began supporting solid elements in stamping simulations as a beta feature. This release delivers significant improvements for this function. These enhancements provide greater accuracy and robustness for forming simulations involving thick or complex blanks.

- Improved Robustness for Solid Element Simulation
- Forming Limit Diagram (FLD) support for solid blanks, including through the thickness of the blank.

Review Simulate a Solid Element Blank to learn more.

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

Figure 3: FLD for a Model with Solid Elements



2.2. Tipping Support for One Step Projects

Tipping is now available for One Step simulations, aligning imported part geometries with the press coordinate system. This improves result interpretation, ensures cross-project consistency, and enhances visualization for identifying potential issues such as undercuts or wrinkling.

Review Tip Your Model to the Press Position to learn more.

2.3. Double Drawbead Now Supported

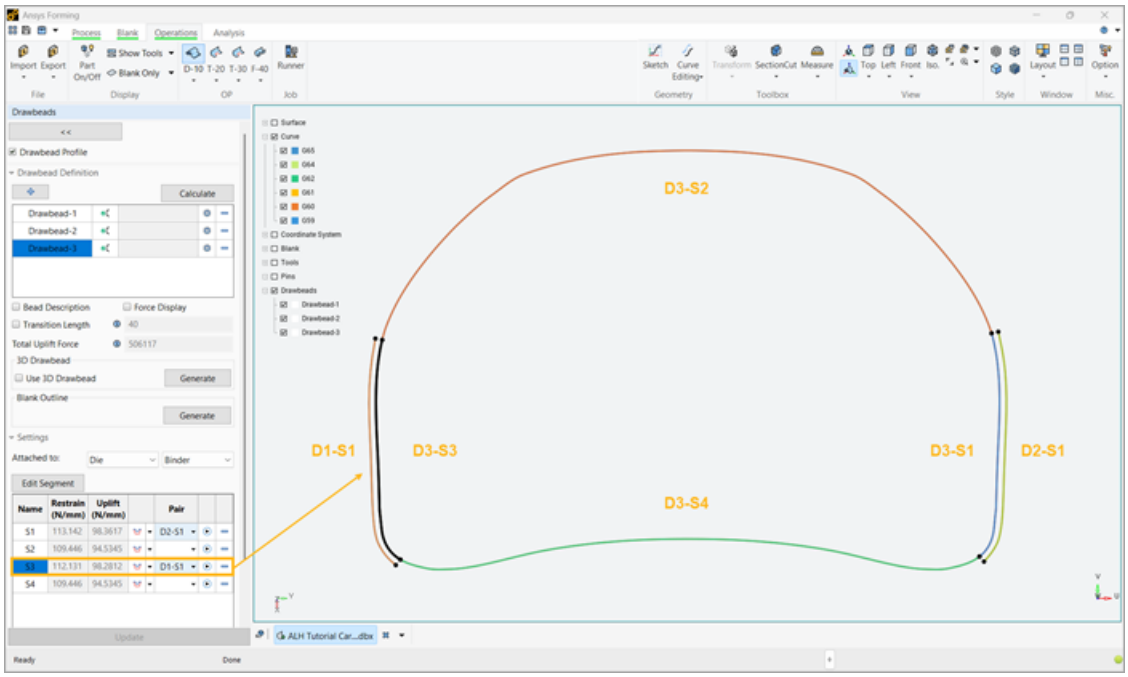
Ansys Forming now allows you to easily generate double drawbead geometry for **Round** and **Round (Advanced)** bead profiles on the binder mesh. The application can then calculate the corresponding force for the paired beads.

Currently, you may only pair beads with the same profile. For example, you may not pair **Round** and **Round (Advanced)** bead types.

Review Add Drawbeads to learn more.

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

Figure 4: Sample Project with Double Drawbeads Defined



The figure above shows a sample project with two double drawbeads defined. The inner curve (Drawbead-3) contains four segments (D3-S1, D3-S2, D3-S3, and D3-S4). The outer curvers to the left and right are defined as Drawbead-1 and Drawbead-2 respectively, each comprised of only one segment. In this model, D3-S3 and D1-S1 are paired to represent a double drawbead (highlighted above), as are D3-S1 and D2-S1.

2.4. Define Multiple Offset Values for Local Tooling Regions

You can now assign multiple offset values to different regions of a single tooling component. This capability allows you to create region-specific tool clearances, such as applying different offset values inside and outside a drawbead on the binder. Use this feature to fine-tune restraining force and better control material flow during forming.

Review Define Multiple Offset Values in the *Ansys Forming User's Guide* to learn more.

2.5. Flexible Gravity Simulation

With improved gravity loading simulation, you now have more flexibility to define the lower support.

Each tool includes an **As Positioning Base** option to maintain stability:

- If only one tool exists, the option is enabled and locked by default.
- If the original positioning tool is present, the option remains enabled and locked.
- If multiple tools exist and none is the original positioning tool, you must select one tool and enable As Positioning Base.

Review Simulate Gravity to learn more.

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

2.6. Operation Suspension in Forming Workflows

You now have greater control when suspending operations in a forming workflow. Previously, suspending an operation also suspended all subsequent operations, which limited flexibility during design testing and iteration.

With this update, you can choose between two suspension options:

- **Suspend:** Disable the selected operation and all following operations.
- **Suspend Current:** Disable only the selected operation while keeping the rest active. You may use this function to skip unfinished or computationally demanding operations, depending on their location in the workflow and impact on subsequent operations.

Review Operations Tab to learn more.

3. Solver & Performance Improvements

New features and enhancements include:

3.1. Enhanced Implicit Solver Performance
3.2. Improved Speed at Binder Closing

3.1. Enhanced Implicit Solver Performance

Implicit solver improvements deliver up to 3X faster performance for gravity loading and springback analysis. Advanced parameters optimize convergence and stability for nonlinear steps, reducing iteration counts and improving robustness.

Review Simulate Gravity and Perform Springback Prediction to learn more.

3.2. Improved Speed at Binder Closing

During the binder closing phase, the solver delays mesh refinement and uses a coarser mesh around the tight radii of 3D drawbeads to improve simulation speed. A finer mesh is applied later in subsequent phases as needed.

4. Postprocessing & Reporting

New features and enhancements include:

4.1. Automatic Report Generation
4.2. New Advanced Solution Type: Stochastic Simulation
4.3. Edge to Drawbead Function
4.4. New Post-Processing Variable: Maximum FLC Failure
4.5. Support for Formability Index with Barlat 2000

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

4.1. Automatic Report Generation

Automatic Report Generation helps you export and communication simulation results in a few simple steps. After completing an analysis, you can generate a structured report that includes a cover page, part and process summary, material details, and operation-specific insights.

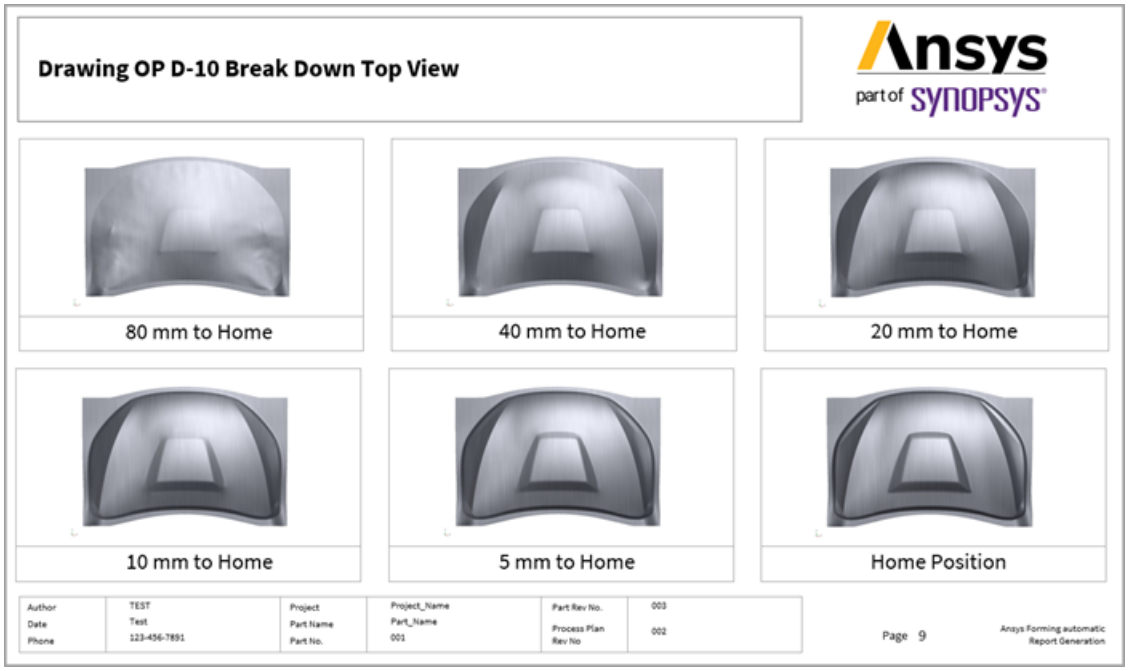
The application exports reports as Microsoft PowerPoint presentations, making them easy to review, present, and share. You can use the standard template or create custom templates to emphasize information most relevant to your projects.

Currently supported results include:

- Gravity simulation
- Drawing operations (formability, thinning, draw-in, and contact information)
- Trimming operations
- Springback compensation
- Measurement operations

Review Additional Tools on the Analysis Tab to learn more.

Figure 5: Sample Slide Generated with Automatic Report



4.2. New Advanced Solution Type: Stochastic Simulation

Stochastic Simulation (robustness analysis) accounts for variability in material properties, boundary conditions, and process settings. This solution type uses Ansys LS-OPT for uncertainty quantification, allowing you to:

- Quantify process robustness under real-world uncertainties.

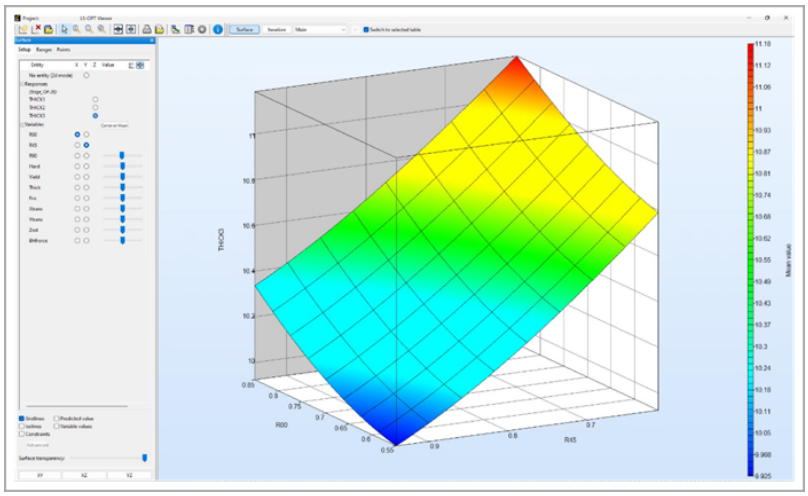
Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

- Identify critical parameters influencing variability.
- Improve design confidence by reducing risk of failure.

An Ansys LS-OPT license is required to use this feature. Access stochastic analysis from the **Process** tab alongside other advanced solution types.

Review Stochastic Simulation to learn more.

Figure 6: Sample Analysis in LS-OPT



4.3. Edge to Drawbead Function

The edge to drawbead function allows you to visualize and measure the distance between the blank edge and the drawbeads throughout the forming process. This function enables you to:

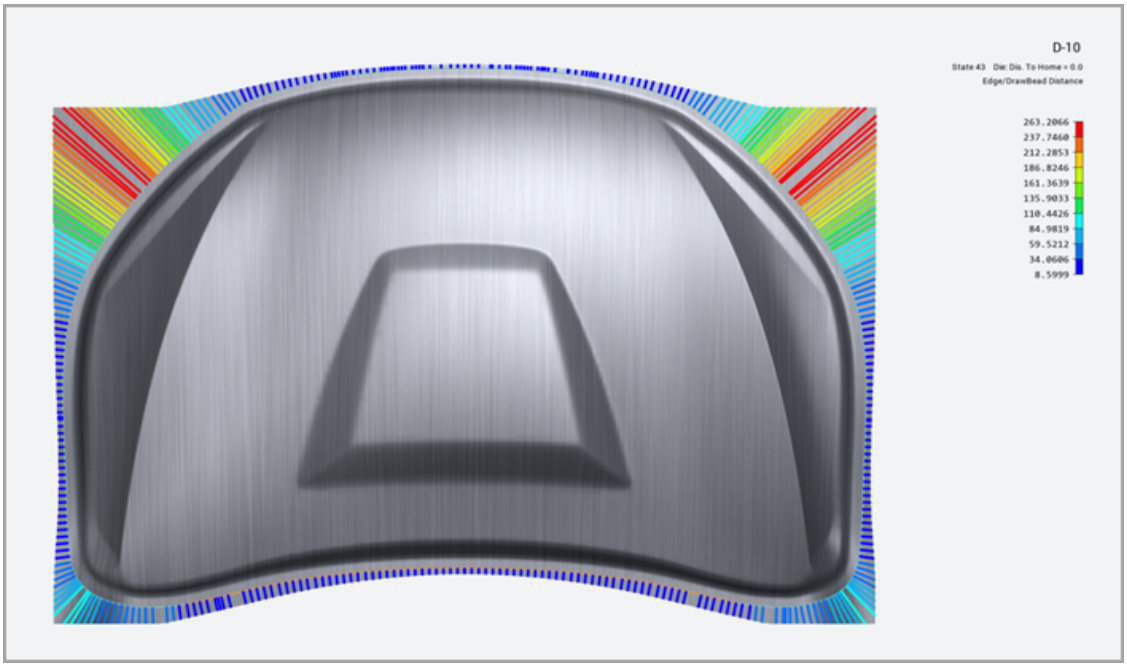
- Ensure the blank edge remains at the intended distance relative to drawbeads, preventing loss of bead engagement.
- Reduce the risk of forming defects such as wrinkling or tearing caused by uncontrolled material flow.
- Optimize drawbead design and binder force settings for improved process robustness.

Access this new capability from the **Analysis** tab, under **Draw-In** drop down menu, and view results as a vector plot with full animation support. This enhancement gives you deeper insight into material flow control, helping you design more reliable tooling and achieve higher-quality stamped parts with fewer iterations.

Review View Edge Draw-In to learn more.

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

Figure 7: Example of an Edge to Drawbead Draw-In Plot



4.4. New Post-Processing Variable: Maximum FLC Failure

Maximum FLC Failure tracks the highest ratio between major strain and the Forming Limit Curve (FLC) strain across all operations, providing a comprehensive view of potential material failure. This variable is currently available only for MAT-3-PARAMETER-BARLAT (MAT36).

Review View Contour Plots to learn more.

4.5. Support for Formability Index with Barlat 2000

You can now use Formability Index (FI) with the Barlat 2000 material model (MAT_BARLAT_YLD2000). This failure criterion considers non-linear strain paths for improved accuracy in forming limit prediction.

5. Documentation

5.1. New Forming Keyword Reference Manual

The release introduces the Ansys Forming Keyword Reference, a manual that catalogues forming-specific LS-DYNA keyword cards used by the Ansys Forming application. The manual organizes forming-specific keyword cards alphabetically and also provides functional groupings that map directly to core tasks such as control settings, contact definition, tool motion, model positioning, and solution strategy.

Use this reference to understand how Ansys Forming interprets your model and to gain deeper insight into solver behavior. Each entry includes a purpose statement, variable definitions with types and defaults, and usage notes. Whether you want to interrogate your output file, modify and run models outside the GUI, or automate your workflows with consistent input formats, this new reference helps you work more efficiently and prepares you for future enhancements to the forming keyword set.

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

Motion Release Notes

The following sections present the new features, enhancements and changes for the Ansys Motion™ standalone Solver, Preprocessor and Postprocessor for the 2026 R1 release.

For Motion in the Workbench application, see the Mechanical application Release Notes.

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

1. New Features
2. Functional Enhancements

1. New Features

The following new features are available:

- **Simulation Configuration Parameters** for the Preprocessor include a new option to execute the Postprocessor automatically when the simulation has ended or stopped.
- **Durability Analysis** now includes a option to allow specification of a **Fatigue** material, either in the **Properties** panel itself or in the **Material Properties**.
- A new **Export Motion Files** dialog has been added to allow selective export of DFS, SSC and DFSXML files, as well as reference and post graphic files.
- The Preprocessor now supports the use of Python code as **Python-Based User Subroutines**.
- FMU export for **FMI** is now available for a Full Vehicle Assembly in the Car Toolkit.
- Areal and δ-F spline **Contact Stiffness** definitions are now supported for multiple contact objects, including **M by N Contact** and links in a **Chained System**.

2. Functional Enhancements

The following functional enhancements are available:

- The user interface for the **Mass** feature in the Preprocessor, used to measure the total mass and mass moment of inertia and center position of selected bodies, has been revised for easier operation.
- **SixMotion** joints have been modified to allow you to check total force, total torque, and total friction results. Additionally, the individual directional components for the six DOFs are reported by derived objects.

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

- **Initial Axis Titles** have been updated for both multi-axis and single-axis **Chart Plots** in the Postprocessor.
- The Body Properties for **Visible Objects** in the Postprocessor has been updated for ease of operation, including new options for **Plasticity Correction**, **DFMF** file path and **Deformation Suppression**.
- Parallel processing efficiency has been improved for analyses where Modal bodies have many connectors.
- The speed of **Nodeset** and **Patchset** creation using a dragged selection in a dfmesh has been improved.
- The name of a **New Curve** generated by a mathematical operation now contains previous curve information as well as math type used.
- Selection of entities is now possible using direct entry of **Node** or **Element** ID.

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

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 specific to the LS-DYNA solver version included in the unified installer, see the R16.1 release notes.
- For release notes specific to changes in the LS-DYNA analysis systems in the Mechanical application, see the Mechanical Application Release Notes.

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
2. LS-PrePost Enhancements
3. LS-Run Enhancements
4. Envyo Application

1. New Documents

The following documents have been added to the LS-DYNA product documentation set in the Help Viewer application. The guides were published on the Ansys website in the previous release and are now available in the Help Viewer:

- **Envyo User's Guide**

  Describes the use and features of the Envyo application, a mapping tool that transfers simulation data between different meshes, primarily for the LS-DYNA product.

- **LS-DYNA Implicit Analysis Guide**

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

2. LS-PrePost Enhancements

The LS-PrePost application includes new features and enhancements at Release 2026 R1 in the following areas:

2.1. Geometry
2.2. Preprocessing
2.3. Postprocessing
2.4. Application
2.5. Solution Explorer

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

2.6. Post Explorer
2.7. General

2.1. Geometry

Geometry updates include:

- The geometry kernel has been upgraded from OCC7.0.0 to OCC7.8.0.
- The CAD assembly model position error, which occurred when translated by HOOPS, has been fixed.
- Geometry tessellation for IGA post models has been optimized.
- When importing an IGA post file (version 1.2), tessellation and fringe is supported.
- The right-click menu in the graphics viewport works for multiple parts. These can be selected by either clicking and dragging (AreaPick) over the required parts, or holding the **A** key and clicking parts when no dialogs are active.
- Additional Python and SCL API for geometry, along with new sample codes, are available.
- Transparency views for geometry parts are supported.
- Measurement values from active geometry tools can be displayed in exported images and animations.

2.2. Preprocessing

Keyword

- **Export YAML file with model include hierarchy**. The following command exports a model's include file tree to a specified filename:

  ```
  cfile command: subsystem inctreeinfo2file filename
  ```

- **Nested Keyword RefBy items**. The items listed in the **RefBy** dialog, which is opened from the Keyword Input Form, have been nested for easy navigation.
- **Keyword Data Filter Table**. The Keyword Data Filter Table allows you to navigate, filter, edit, display, and compare keyword data. It enables efficient handling of large and complex LS-DYNA models.
- **Draw keyword entities**. Toggle the eye icon in the first column to draw the keyword if it is drawable.
- **Right-click menu**. You can select items in the Keyword Data Filter Table and then right-click to open a contextual menu of further options.

Entity Display

- **Improved Entity Display**. *BOUNDARY_SALE_MESH_FACE boundaries can be drawn with different colors, and segments can be drawn on boundaries that are non-reflective.

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

Measure

- **Optimize segment to segment measuring**. In the **Measure** dialog, when **Item** is set to **Dist Segm to Segm**, you can select the **Optimize** check box to optimize the algorithm when the nearest segment between Group 1 and Group 2 (in either direction) must be in the direction of the segment normal. This setting is helpful when solid elements represent the blank part for metal forming analyses.

IGA

- **Support *BOUNDARY_PRESCRIBED_MOTION_POINT_UVW in Entity Creation dialog**. In the **Entity Creation** dialog, under **Boundary** > **Prescribed Motion (BPM)**, the keyword option POINT_UVW is available. This is used when prescribed boundary conditions are applied to IGA shells.
- **Removed IGA_NURBS_2D_XYZ and IGA_FACE_XYZ entities**. Removed the IGA_NURBS_2D_XYZ and IGA_FACE_XYZ entities in trimmed NURBS 3D entity for cube linear mapping. Node(control point) was removed.
- **Create trimmed IGA solid from STEP file**. You can create a trimmed IGA solid from a STEP file automatically by enabling **Create(Auto)** when importing the `.stp` geometry model and clicking **Apply**. Every solid part will be converted to trimmed IGA solid respectively.

  Note:

  This feature is supported for `.stp` files only.

2.3. Postprocessing

Fringe Component and Fringe Legend

- **Shell thickness visualization (PostDB)**. You can display fringe contours on shell elements while Shell Thickness is on. This allows for a more realistic view of stress/strain distribution across the geometry.
- **Strain metrics**. For **Strain** components, there is support for **Tresca (max shear strain)** for solid elements.
- **Fringe of max/min IPT selection of *INITIAL_STRESS with multiple IPT for solid elements** . *INITIAL_STRESS keywords that have multiple integration points are supported in the **Max**/**Min** integration point selection option of the Fringe Component dialog.
- **Laminate shell results enhancements**. For **LSDA** defined fringe components, you can load eloutdet files using the **Branch** drop-down menu to extract and fringe strain data across different integration points in laminate shells.

  There is also a new **Option** drop-down menu to switch the result coordinate system between **d3plot** (Global system) and **element** (Local system).
- **Fringe legend menu options**. From the right-click menu of the Fringe legend, the following options have been added:

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

Identify Min/Max. Highlights the element or node with the minimum or maximum value in the model view.

New Legend. Defines a new Fringe legend (**Change Legend** > **New Legend**). From here you can define individual legend colors, the overall color palette, values, and fringe limits.

Palette Colors. Sets the legend to a pre-defined color palette. You can select from Default, Turbo, Cool-warm, Plasma, Inferno, Viridis, and User File.

- **Local Fringe legend support in split windows**. User-defined Fringe legends can be used across split windows by right-clicking the legend and going to **Change Legend** > **Select Legend**. Across split windows, legends can vary in color, fringe level, font size, and placement within the display.

ASCII

- **Node classification by Set ID**. In **Post** > **ASCII**, you can classify Node IDs in the bndout branch based on their specific Set ID. To do this, select the **Highlight** check box and then, in the Set Node dialog, select **By Set/Grp**. This helps find and select specific nodes to Plot within large datasets.

- **Rotational curve values unit display**. You can switch the display of rotational curve values between degree and radian (default) from the **Settings** tab in LS-PrePost, under **Configuration Settings** > **Display** > **General** > **Angular System**. This option influences BINOUT and ASCII curve plots of rotational-displacement, rot-velocity and rotational acceleration (nodout/rbdout).

BINOUT

- **Element classification by Set ID**. In **Post** > **Binout**, you can classify element data based on their specific Set ID. To do this, select the **Highlight** check box and then, in the Set Elem dialog, select **By Set/Grp**. This helps find and select specific elements to Plot within large datasets.

Vector Plot

- **Velocity vector plotting for D3DAT**. You can plot velocity vectors for `d3dat` files.

3D Graph View

- **3D Rainflow visualization**. Added support for Rainflow Metric plots within the 3D Graph view.

2.4. Application

In the **Application** menu, the following enhancements have been made to **Fragment Analysis** and **Occupant Safety**:

- **Fragment Analysis for SPH**. The Fragment Analysis tool supports Smoothed Particle Hydrodynamics (SPH).

- **Dummy Positioning enhancements**. In **Occupant Safety** > **Dummy Positioning**, the following buttons have been added:

Show dummy parts only. Displays only the parts referenced in the Dummy tree in the model view.

Run Dyna. Creates discrete elements (*ELEMENT_DISCRETE) on positioning nodes and uses the LS-DYNA simulation to pull the Human Body Model (HBM) to a target posture.

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

2.5. Solution Explorer

General

- **Keyword support for reading object data**. LS-PrePost can decode the following keywords for reading different object data:

  Pressure. *LOAD_SHELL_ELEMENT, *LOAD_SHELL_SET, *LOAD_SEGMENT

  SPC. *BOUNDARY_SPC_NODE, *BOUNDARY_SPC_NODE_BIRTH_DEATH

  Force/Moment. *LOAD_NODE_SET, *LOAD_NODE_POINT

  Prescribed Motion. *BOUNDARY_PRESCRIBED_MOTION_NODE

  RBE3. *CONSTRAINT_INTERPOLATION

- **Support composite layers in Shell Part objects**. You can add composite layers in Shell Part objects. The following keywords can be decoded within the model tree: *PART_COMPOSITE, *PART_COMPOSITE_CONTACT, *PART_COMPOSITE_LONG, and *PART_COMPOSITE_LONG_CONTACT.

- **Support discrete elements and section properties in Discrete Part objects**. You can add discrete elements and section properties in Discrete Part objects.

- **Support Mass Element objects in Parts tree**. You can add mass element objects in the Parts tree.

- **Support Simple Airbag Models objects in Airbags object**. You can add the new Simple Airbag Models object in the Airbags object.

EM

- **Support cell connections among replicated Unit Cells**. You can add cell connections among replicated unit cells by right-clicking the **Unit Cell** from the Solution Explorer tree and selecting **Cell Connection**. From the Property Explorer you can then set the **Connection Type**, **Connected Isopotentials**, and **Connected Cells**.

Material Library

- **Support Plastic materials**. You can define plastic materials (compression, tension) by right-clicking the **UserDefined Database** and selecting **New Material Plastics**.

- **Support Rubber materials**. You can define rubber materials (simplified rubber/foam) by right-clicking the **UserDefined Database** and selecting **New Material → Rubber**.

- **Support constitutive equations for metal materials**. You can add new constitutive equations - such as Johnson Cook, Power Law, Plasticity with Damage, and so forth - for metal materials in the UserDefined Database.

- **Support *MAT_MODIFIED_PIECEWISE_LINEAR_PLASTICITY in UserDefined Database**. *MAT_MODIFIED_PIECEWISE_LINEAR_PLASTICITY is supported in the UserDefined metal materials.

- **New Structural Behavior options for Metal objects**. In the UserDefined Database, Structural Behavior options for Metal objects include **SpotWeld** (*MAT_SPOTWELD_) and **SpotWeld Daimlerchrysler** (*MAT_SPOTWELD_DAIMLERCHRYSLER_).

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

- **Support Spring and Damper materials**. You can define Spring and Damper materials by right-clicking the **UserDefined Database** and selecting **New Material** → **Spring Damper**.
- **BaoSteel material support**. You can add materials from the **BaoSteel** folder in the **Built-in Database**.
- **lspp_forming_temp\lspp_matlib_UnitB folder materials**. You can add materials from the `...\lspp_forming_temp\lspp_matlib_UnitB` folder, which is located in the LS-PrePost installation path, in the **Built-in Database**.

2.6. Post Explorer

Fringe

- **Moved Fringe legend settings**. All Fringe legend settings have been moved to the Fringe legend right-click menu.
- **Record Plot Mode for selected parts**. You can record the **Plot Mode** for selected parts under a fringe component object. To do this: select the parts from the tree, right-click, and then select **Plot Mode** followed by the required mode (for example, **Shaded**). From the Post Explorer tree, right-click the Components folder and select the required results view.

BINOUT

- **New BINOUT branches**. Post Explorer supports the **matsum** and **sbtout** binout branches.
- **Loading the binary output**. LS-PrePost supports loading the Binout, which will be searched for in current working directory.

2D Graph Viewer

- **Integration with 2D Graph Viewer**. Post Explorer States are supported in the 2D Graph viewer. Right-click the viewport and go to **Plot Manage** to open the view state in the 2D Graph Viewer.
- **Improved point selection**. Curve points that are selected in the 2D Graph menu are also displayed in the main window's XY Plot area.

Properties

- **Model Title options**. You can define the model title using the **Title Option**. Options include:

  By D3Plot. Sets the title based on text found in the *TITLE keyword.

  By Working Dir. Sets the title based on the name of the folder where the results are saved.

  User Defined. Allows you to enter a title manually.

2.7. General

General updates to LS-PrePost include:

- **SCL from the command line**. You can run SCL from the command line.

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

- **Latest FEMUNZIP libraries**. The latest FEMUNZIP libraries (version 17.1.901) are supported.
- ***SET_PART_TREE for FEMZIP models**. FEMZIP models containing *SET_PART_TREE information in the Subset Explorer are supported.

3. LS-Run Enhancements

The following enhancements for LS-Run are available in release 2026 R1. Refer to the product-specific documentation for full details.

- **Core Pinning**. From the **Settings** dialog, you can pin each LS-DYNA process to a unique core ID when running shared memory parallel processing (SMP) or massively parallel processing (MPP) with Intel MPI on a local machine. This typically increases analysis performance and is beneficial if, for example, the job is run on a computer with both E-cores and P-cores, as you can specify that LS-DYNA processes run on P-cores only.

4. Envyo Application

The Envyo application includes new features and enhancements at Release 2026 R1 in the following areas:

4.1. ACP-to-DYNA Mapping Option Enhancements
4.2. New Workflow Examples
4.3. Performance Improvements

4.1. ACP-to-DYNA Mapping Option Enhancements

The ACP-to-DYNA mapping option includes the following new material keywords and commands:

Material Keywords

- MAT_ELASTIC (MAT_001)
- MAT_HONEYCOMB (MAT_026)
- MAT_LOW_DENSITY_FOAM (MAT_057)
- MAT_FU_CHANG_FOAM (MAT_083)
- MAT_MODIFIED_HONEYCOMB (MAT_126)

Commands

- **ConvertTShellsToSolids** – Converts single-layered solid elements to `*ELEMENT_SOLID_ORTHO` keywords.
- **LinearizeQuadraticSolids** – Removes all second-order nodes of quadratic solids, and exports elements as linear solids in the generated keyword file.

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

4.2. New Workflow Examples

The following workflow examples have been added:

Continuous Fiber

- Shell to Shell: Single-Layered Draping
- Shell to Shell: Multi-Layered Draping
- Shell to Shell: Multi-Layered Draping with Two Fibers per Integration Point
- Plybook to Shell: Mapping Ply Orientations from *ELEMENT_SHELL_BETA to a Single-Layered Shell

Image Processing

- Grayscale to Shell: Mapping Image-Based Clusters to a Finite Element Model
- Grayscale to Solid: Mapping Adhesive Wetting Damage Parameter to Integration Points
- Point Cloud to Shell: Mapping Temperature Data to Nodes

4.3. Performance Improvements

Addressed bottlenecks reduce processing time by 30–60%.

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

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. 131)
- CFX (p. 153)
- CFD-Post (p. 155)
- TurboSystem (p. 157)
- TurboGrid (p. 159)
- BladeBuilder (p. 161)
- Polyflow Classic (p. 163)
- Forte (p. 165)
- Ansys Chemkin (p. 167)
- FENSAP-ICE (p. 169)
- EnSight (p. 171)
- Rocky (p. 181)
- Freeflow (p. 175)

Fluent Release Notes

The following sections contain release information for Ansys Fluent 2026 R1.

1. Supported Platforms for Ansys Fluent 2026 R1
2. New Features in Ansys Fluent 2026 R1
3. Updates Affecting Ansys Fluent 2026 R1 Code Behavior
4. Features To Be Removed in a Future Release

Backwards Compatibility: Ansys Fluent 2026 R1 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 2026 R1

Information about past, present, and future operating system and platform support is viewable via the Ansys website.

2. New Features in Ansys Fluent 2026 R1

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 2026 R1 are listed below.

Fluent Guided Meshing Workflows

- The **Watertight Geometry** Workflow
  - Stairstep options for thin volume meshing have been deprecated.
  - The parallel region compute option for conformal regions has been added to the Generate the Surface Mesh task, resulting in improved performance for cases with many objects.
  - The following additional controls are now available in the Extrude Volume Mesh task for extruding periodic zones:

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

→ Extend to Periodic Pair

→ Extend Periodic

→ Max Layer Height

  - In the Add Local Sizing task, the Ignore Proximity Across Objects option has been added for controlling proximity size using faces only.
  - Autodetection of thin regions has been improved for thin volume meshing. Auto-separated source and target zones created during thin volume detection are now automatically merged at the end of volume meshing, resulting in fewer zones and boundary condition settings in the solver compared to previous releases.

- The **Fault-tolerant Meshing** Workflow
  - The CAD Assemblies capability has been deprecated and will be removed in a future release. For similar capability consider using the Part Management capabilities.
  - In the Add Boundary Layers task, label(s) that have already been assigned boundary layers will no longer be available in the list of named label(s).
  - placeholder

- The **2D Meshing** Workflow
  - The Axisymmetric Sweep task has been added to the workflow and can be used to create a 3D mesh by revolving a 2D axisymmetric mesh about a specified axis.

Mesh Generation

- When using curvature size functions with the Rapid Octree mesher, the following are now supported as full features:
  - You now have the ability to change the criterion used for the refinement of the curved surfaces. The new choices—`switched` (which is now the default) or `arc-estimate`—are less sensitive to surface faceting and can provide refinement even as the facet resolution increases compared to `facets-normal-angle` (the default in previous releases). If you want to revert to the criterion used in previous releases in order to maintain the same cell counts, you can use the following text command: `mesh/rapid-octree/mesh-sizing/curvature-refinement-options/criterion facets-normal-angle`
  - You can now set maximum and minimum angle thresholds between the two facets for angular refinements with curvature size functions; such settings can be useful, for example, to exclude sharp corners and to prevent spurious refinements, respectively.

  For further details, see Creating Size Functions.

2.2. Solution Mode

New features available in the solution mode of Ansys Fluent 2026 R1 are listed below. Where appropriate, references to the relevant section in the User's Guide are provided.

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

User Experience

- *Search* is now available for filtering contents in both Preferences and the Console.
- The Raytracing Display window, where realistic images and volume renders are displayed, now accepts graphics manipulations such as pan, rotate, and zoom. Refer to Realistic Rendering Using Raytracing in the *Fluent User's Guide* to learn more about these types of displays.

Files

- You can now import and append TurboGrid mesh files (`.gtm`, `.def`).

Third-party Software

- HOOPS is now updated to 30.14 from 27.23 to address security vulnerabilities.

Solver-Meshing

- When using overset meshes, the Green-Gauss Node Based gradient scheme is now supported with both the pressure-based solver and the density-based solver. (Overset Meshing Limitations and Compatibilities)
- When using a mesh interface with the Green-Gauss Node Based gradient scheme, a new expert option is available that allows you to control the enforcing of bounds on interpolation weights for solution variables at the nodes of non-conformal interface boundaries. You can enter the following text command to disable the clipping of the node interpolation weights, which can improve the quality of gradients at the cells adjacent to the non-conformal interfaces: `solve/set/gradient-options/green-gauss-node-based/weight-treatment-at-sliding-boundary? yes`.
- The following improvements are available when you create a non-conformal mesh interface using the default one-to-one interface creation method:
  - A list is now automatically printed in the console for any associated boundary zone that is deemed to be over-intersected (that is, a boundary zone that has at least one face in which the total intersection area for the face is bigger than the area of that face multiplied by a factor of 1.25), along with metrics that indicate the extent of the over-intersection. This information is also printed if you list the details of an existing mesh interface. You should investigate all over-intersected boundary zones, to confirm if their mesh interfaces are properly defined.
  - You can now specify whether the lists of mesh interfaces printed in the console are ordered by their overlap area (that is, the area that the two boundary zones overlap) or by their overlap area percentage (that is, the overlap area divided by the area of each of the two boundary zones). Previously the lists were always ordered by the overlap area percentage.
  - You can now define a minimum area of overlap that is used to create mesh interfaces when zones are paired using the automatic method. This can help you avoid unnecessary or unintentional mesh interfaces. Previously you could only define a minimum overlap area percentage.
  - When you are deleting existing mesh interfaces that have small overlap, you can now specify an area that is used for deletion: if the area that the two boundary zones overlap for any

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

mesh interface is less than this value, then it will be deleted. Previously you could only specify an overlap area percentage for deletion.

– The non-conformal interface surfaces from one-to-one mesh interfaces now participate in postprocessing operations. Note that for the non-overlapping face zones, the faces adjacent to the intersected zone will only be displayed or contribute to integrals if they are sufficiently large compared to their parent faces (and then the parent faces will be used).

For details about non-conformal mesh interfaces, see Using a Non-Conformal Mesh in Ansys Fluent.

- The following improvements are now available when deleting or deactivating cell zones:

  – You can now delete or deactivate a cell zone with boundary zones that are non-manifold. This was previously possible in the meshing mode of Fluent and is now also supported in the solution mode. This can be useful when postprocessing for example, as you can visualize a single small zone by deactivating all of the other zones.

  – Fluent now robustly handles attempts to delete or deactivate a cell zone with a boundary zone that has misoriented faces; while such an action has never been and is still not fully supported, it will no longer result in a broken mesh or the abnormal termination of Fluent, and this allows you proceed using Scheme commands at your own risk (as noted in warning messages in the console).

  For details about deleting and deactivating cell zones, see Deleting Zones and Deactivating Zones, respectively.

- It is now possible to extract cells that are marked by a specified cell register (as well as any specified additional layers of neighboring cells), so that they are preserved and all other cells in the mesh are deleted. Extracting cells can be useful when working with a large mesh that is difficult to manipulate, as it can allow you to focus on a small region of interest and easily perform actions such as postprocessing. (Extracting Cells)

- You can now easily fuse all of the contacting face zones between selected cell zones, rather than having to select all of the individual face zones. This is particularly useful if you have disconnected cell zones and want to reestablish connection. (Fusing Face Zones of Selected Cell Zones)

- The ability to project the nodes from one or more boundary zones so that they are moved to be coincident with an auxiliary geometry definition is now supported as a full feature. This can be useful for sliding mesh simulations, as you can make sure that all of the nodes of a boundary pair for the non-conformal interface are on the same plane or on the same surface, in order to reduce simulation errors. (Projecting Boundary Zones Onto an Auxiliary Geometry Definition)

Cell Zones and Boundary Conditions

- When using the pressure-based solver with a turbulence model, a treatment option is now enabled by default, such that the effects of turbulence wall functions are removed at walls where you have specified zero shear. This treatment applies the Neumann boundary condition for the turbulence quantities, so that the walls behave as symmetric boundary conditions. This enhances the convergence, as no turbulence wall modeling is performed. It is possible that this treatment may make minor changes to the solution near the wall, but they should not be

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

meaningful. You can disable this treatment by entering the following text command: `solve/set/advanced/specified-zero-shear-alternative-treatment? no`. (Specified Shear)

- When using the pressure-based solver, a new treatment is applied by default to the following boundary conditions, in order to avoid inaccurate flow prediction when the direction specified is not well aligned with the local boundary face normal:

  – pressure inlets with **Direction Vector** selected from the **Direction Specification Method** drop-down list (see Pressure Inlet Boundary Conditions)

  – pressure outlets that: have the **Prevent Reverse Flow** option disabled; have either **Direction Vector** or **From Neighboring Cell** selected from the **Backflow Direction Specification Method** drop-down list; and experience backflow conditions (see Pressure Outlet Boundary Conditions)

  If you do not want to take advantage of this increased accuracy, you can undo the default treatment for pressure inlets and pressure outlets with misaligned directions using the appropriate prompt of the following text command: `solve/set/previous-defaults/undo-2026r1-default-changes?`.

Heat Transfer / Radiation

- You can now model ambient radiation to the external environment without needing to create a fluid box around the geometry. This significantly simplifies the case set up by eliminating the need to mesh/model the fluid medium between surfaces and can also significantly reduce computation time. (Ambient Radiation Modelling)

- The Macro heat exchanger model now supports non-conformal interfaces at flow inlets and outlets.

Turbulence

- The original version of the GEKO model (GEKO_2015) has shown some conceptual weaknesses which have motivated a moderate re-formulation (GEKO_2024) in order to improve tuning capabilities. The main issue was the need to separate the tuning coefficients of the model between coefficients affecting boundary layers and coefficients affecting frees shear flows.

  The improved GEKO_2024 is the default GEKO model version in 2026 R1. The original GEKO_2015 is still available, but the recommended model version is GEKO_2024.

  Note that in `.cas` files created with Fluent versions previous to 2026 R1 which not have GEKO enabled, the default GEKO version will be set to 2024 in the background and version dependent coefficients (CMIX, CJET and CJET_AUX) will be adjusted to their default values.(Generalized k-ω (GEKO) Model)

- For the WALE LES model, the turbulence length scale is now limited close to the walls for high aspect ratio meshes. This improves solver stability for cases with poor mesh quality and without influencing the solution and separation prediction on isotropic meshes. (Wall-Adapting Local Eddy-Viscosity (WALE) Model)

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

- When using the GPU solver with the LES model, for near-wall treatment, you can now have the input fields used for wall-function evaluation be taken from the second cell off the wall. (Features Supported by the Fluent GPU Solver)

- When using the GPU solver with either the kω-SST or GEKO models, the tabulated **Near-Wall Treatment** is now available in addition to the correlation method. (y+-Insensitive Near-Wall Treatment for ω-based Turbulence Models)

Turbo Setup Workflow

- Performance gains have been notably improved when copying and merging multiple cell zones in the Turbo Setup Workflow.

Discrete Phase Model

- For simulations that involve the Lagrangian wall film (LWF), a new caloric temperature treatment is now available for the film temperature calculation. The treatment provides improved accuracy in cases where:

  – the particle material has temperature-dependent specific heat

  – the film consists of two or more materials with different physical properties that coexist on the same wall face

  For more information, see Calculation of Wall Film Face Variables in the *Fluent Theory Guide*.

- In coupled simulations with the GPU solver, the **Linearize Source Terms** option is now fully supported for species sources.

- In simulations with the GPU solver where particle species are evaporating, the binary diffusivity of the particle material can now be modeled using the film-averaged approach.

- For cases involving the film-to-wall heat transfer model, you can now specify the distance between the film starting position and the heat transfer region. This capability can significantly improve the accuracy of convective film heat transfer simulations, particularly when the film start point is upstream of the heat transfer region. (Setting the Film-to-Wall Heat Transfer Model)

Multiphase Models

- A text command, `solve/set/multiphase-numerics/interphase-interactions/turbulent-dispersion/disable-turbulent-dispersion-next-to-flow-boundaries?`, is now available for disabling momentum sources due to turbulence-dispersion in cells next to flow boundaries. In certain cases this can significantly improve convergence.

- For Eulerian multiphase cases that involve the critical heat flux (CHF) boiling model, droplet diameters for the primary phase can now be modeled as a constant or using the bending diameter model. (Procedure for Setting the Boiling Model in the *Fluent User's Guide*)

- For the wall boiling model, a new text command, `solve/set/multiphase-numerics/boiling-parameters/bubble-diameter-model-options`, has been added. The command allows you to select between the default and Unal bubble diameter modeling options. This functionality was accessed through a scheme command in previous releases.

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

- For multiphase species-transport simulations, a vanishing phase treatment has been introduced. The treatment respects initial mass fraction values in cases where phase change depends on partial pressure or where species transfer occurs between phases. (Modeling Multiphase Species Transport)

Electric Potential Field and Electrochemistry Model

- The **Electrolysis and H2 Pump Model** has been renamed **Electrochemical Devices** and has been enhanced with the following capabilities:
  - The following new device types have been introduced:
    - → Solid oxide fuel cell (SOFC)
    - → Solid oxide electrolysis cell (SOEC)
    - → Customized electrochemical reactions

    See Electrochemical Devices in the *Fluent Theory Guide* for details.
  - The ability to account for species diffusion through the membrane of electrochemical devices has been added. (Setting Advanced Options (Advanced Tab))
  - In the unresolved electrochemistry model, you can now customize the membrane electrolyte conductivity via the `DEFINE_ELECTROLYSIS_COND` user-defined function (UDF). For details, see `DEFINE_ELECTROLYSIS_COND` in the *Fluent Customization Manual*.
- For the Lithium-ion battery model, the following additional postprocessing quantities are now available in the **Lithium...** category:
  - **Overpotential in Li-Intercalation Reaction**
  - **Overpotential in SEI Growth Reaction** (aging model)
  - **Overpotential in Li-Plating Reaction** (aging model)

  For definitions of these field variables, refer to Alphabetical Listing of Field Variables and Their Definitions in the *Fluent User's Guide*.

Battery Model

- The following battery modeling capabilities are now supported with the GPU solver:
  - **CHT Coupling** solution method
  - **Joule Heat in Passive Zones** energy source option
  - Battery ROM tool kit
- For the Newman's P2D model, you can now import model parameters from a BPX-format `JSON` file using the BPX file reader. (Inputs for the Newman's P2D Model)
- When using the ROM tool kit, you can now specify a heat flux distribution on selected walls (Create ROM Training Data (ROM Data Creator Tab))

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

- For the one-equation thermal abuse model, the parameter estimation tool is now available in the graphical user interface (GUI). Consequently, the **Standalone Thermal Abuse Model** dialog box has been renamed to **Fitting Tool/Standalone Thermal Abuse Model**, and the **Preview Thermal Abuse Model...** button in the **Battery Model** dialog box (**Advanced** tab) has been renamed to **Fitting Tool/Standalone Model...**. (Fitting Tool/Standalone Thermal Abuse Model)

Proton Exchange Membrane Fuel Cell (PEMFC) Module

- The degradation model to simulate degradation processes in the membrane is now available. (Setting Material Chemical Degradation)

Solver

- When using the Green-Gauss Node Based gradient scheme, the improved treatment of symmetry and periodic boundaries in the node-based reconstruction gradient is now available for the pressure-based solver. This treatment ensures that the gradient respects symmetry and periodic constraints at the discrete level, thus better matching results obtained on an untruncated domain. (Green-Gauss Node-Based Gradient Evaluation)

- When using the pressure-based solver, you can now use a user-specified gradient scheme to compute the reconstruction gradient that is used in the second-order pressure interpolation. This allows better accuracy of the second-order scheme compared to the current default (which is based on the Green-Gauss Cell Based scheme for general unstructured meshes). To use the user-specified gradient scheme, enter the following text command: `solve/set/gradient-options/pressure-reconstruction-type? yes`.

- When using the Least Squares Cell Based gradient, you can now control the weighting factor used in the assembly of the least squares matrix system that computes the gradient weights. You can specify that the weighting factor is unweighted, inverse distance weighted, or the default inverse distance squared weighted. The use of unweighted or inverse distance weighted may result in improved accuracy, especially at physical boundaries. To specify the weighting factor, use the following text command: `solve/set/gradient-options/least-squares/weighting-factor`.

- When using the pressure-based solver, you can now apply the PRESTO! pressure interpolation treatment at fluid physical boundaries. This is intended to be used with any other pressure interpolation scheme besides PRESTO!, in order to improve the flow prediction without being subject to the mesh size sensitivity of the PRESTO! scheme. (Choosing the Pressure Interpolation Scheme)

- You can now specify the use of the bi-conjugate gradient stabilized method (BCGSTAB) for the energy equation in all future sessions of Fluent by enabling the **BCGSTAB for Energy Equation** option in the **Simulation** branch of the **Preferences** dialog box (accessed through **File/Preferences...**). This can be helpful with conjugate heat transfer (CHT) problems, as described in Conjugate Heat Transfer. For further details about the BCGSTAB method, see Setting the Stabilization Method.

Fluent Native GPU Solver

- The following improvements are available when using expressions for report definitions and unsteady statistics with the GPU solver:

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

  - Additional fields are now supported as a full feature for fully supported models, including fields for material properties, turbulence modeling, reaction and combustion, the volume of fluid (VOF) model, and user-defined scalars (UDS). For the full list of supported fields and other details, see Using Expressions with Report Definitions and Unsteady Statistics.

  - When creating an expression report definition, the selection of existing report definitions as part of the definition is now supported. For further details, see Using Expressions with Report Definitions and Unsteady Statistics.

- When using the GPU solver, CPU/GPU remapping is now supported with asynchronous (async) outputting, as long as the case does not have a command defined through the **Execute Commands** dialog box for execution during the calculation that requires downloaded data from the GPU. For further details about these features, see CPU/GPU Remapping and Asynchronous Outputting, respectively.

- When using the GPU solver, asynchronous (async) outputting is now supported when automatically exporting solution data as EnSight DVS files during a calculation. For further details about these features, see Asynchronous Outputting and Exporting Solution Data as EnSight DVS Files, respectively.

- The GPU Solver can now be started in hybrid precision, resulting in improved performance and memory consumption when compared to the double precision solver.

- Supersonic inflow (ideal gas only) is now supported at velocity inlets and mass flow inlets.

- For the Potential/Electrochemistry model, time-dependent values and contact resistance can now be specified at wall boundaries.

- Vertex Average, Vertex Minimum, and Vertex Maximum monitors can now be defined on zone, point, and plane surfaces.

- Monitors can now be defined on iso-surfaces.

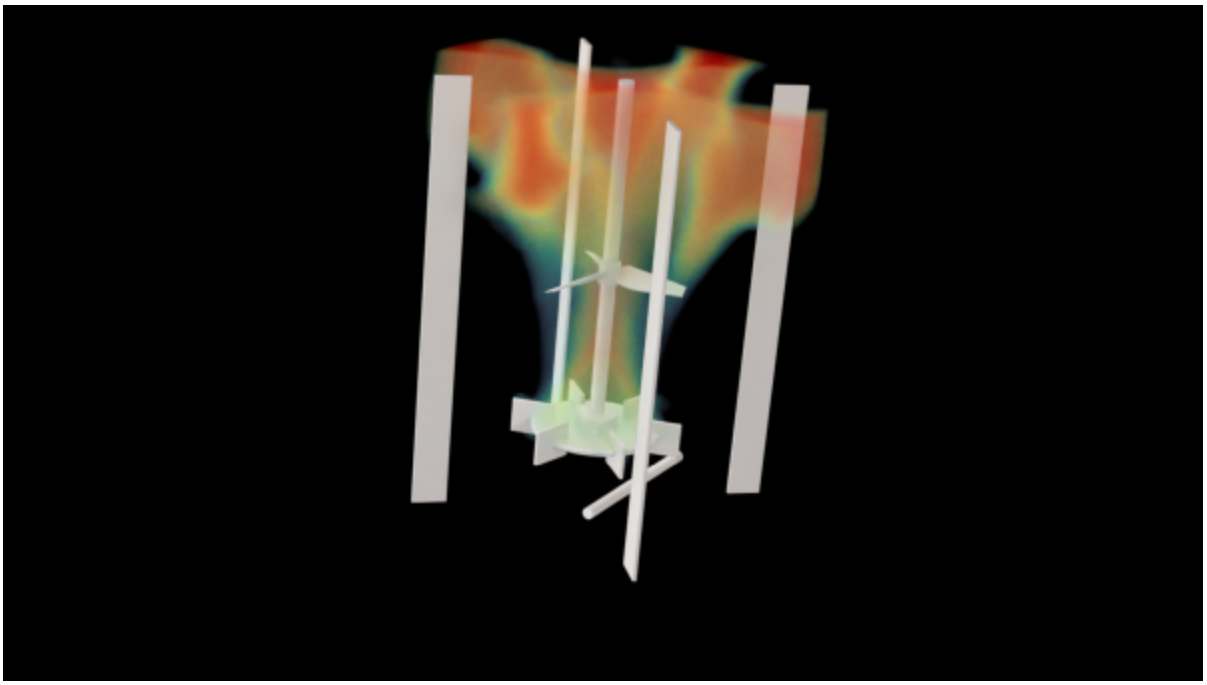
Graphics, Reporting, and Postprocessing

- You can create key frame-based animations without having a case or data file loaded by using HSF animation files. Refer to HSF File and In Memory Key Frames in the *Fluent User's Guide* for additional information.

- Volume rendering is now full feature, allowing you to create images like Figure 8: Example Volume Rendering with Mesh (p. 140). These volumes can also be included within animation definitions. Refer to Displaying Volume Renderings in the *Fluent User's Guide* for additional information.

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

Figure 8: Example Volume Rendering with Mesh



- You can now create new and access existing graphics objects (such as contours, vectors, pathlines, and so on) prior to reading a data file or initializing the flow field.
- The import of STL surfaces is now done through the **External Surface** dialog box. This new approach (previously import was done using the **Imprint Surface** dialog box) uses a better algorithm that results in superior data mapping. Refer to External Surfaces in the *Fluent User's Guide* for additional information.
- You can now split surfaces, such as planes or external surfaces, at each location where that surface crosses a boundary, allowing for greater granularity in postprocessing. Refer to Split Surfaces in the *Fluent User's Guide* for additional information.
- The iso-surface and iso-clip algorithm is improved.
  - For iso-surfaces, it offers improved performance, particularly when creating multiple surfaces at once. It also has improved handling of boundaries during surface creation.
  - For iso-clips, faceting is improved.

Adjoint Solver Module

- The prescribed profile boundary condition within the design tool can now be defined using an expression with multiple input parameters.
- The Export Cad dialog box can now be accessed within the design tool, enabling export of a modified geometry as a CAD file for use in other CAD software packages.

Fluent for Arm

- The following features are now supported with Fluent for Arm:

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

  - Sundials in reacting flow models
  - Fluent native GPU solver with NVIDIA GPUs

  For more details, see Running Ansys Fluent on Arm Compute Nodes.

Parametric Studies

- With the ability to run concurrent design point sessions (either locally or remotely) directly from the parametric study, the need to save journal files for submission to a job scheduler or to run the journals in different Fluent session(s) is no longer required. Therefore, the **Save Journal** option(s) has been removed from the graphical user interface (that is, the ribbon and the corresponding context menu). The functionality remains available in the text user interface, but will be deprecated in a future release.
- In addition to configuring remote cluster job submission details for concurrent parametric design point studies, you can now submit your remote concurrent parametric jobs directly to Ansys Cloud Burst (see Configuring Remote Concurrent Parametric Sessions for details).

Simulation Reports

- You can choose to manage your data and generate your simulation reports using a local server (the default), or by directly updating a ("serverless") database with the **Ansys Dynamic Reporting Workflow**. You can choose the more direct approach using preferences (under the **Simulation Reports** heading of the **Simulation** category in the **Preferences** dialog box

Cloud Computing and Remote Resources

- When specifying journal file details while submitting a remote or cloud-based job submission, you can now automatically generate a simulation report as either HTML, PDF, or as a PowerPoint presentation (see Specifying Journal Details for Remote or Cloud Jobs for details).
- From within your Ansys Fluent session, you can now choose to submit a single Fluent job to any HPS-supported remote cluster (see Performing Calculations Using Remote Resources for details).
- Improvements have been made when managing and the viewing the details of jobs submitted to Ansys Cloud Burst or to other remote resources, such as being able to rerun jobs or manually refresh job details (see Viewing and Managing Existing Remote or Cloud Jobs Within Your Ansys Fluent Session for details).
- Improvements have also been made when viewing and managing existing jobs on a remote cluster or cloud-based resources (see Viewing and Managing Existing Remote or Cloud Jobs Within Your Ansys Fluent Session for details).

Cell Registers

- It is now possible to visualize a cell register in the context of the rest of the mesh using the **Contours** dialog box. (Overset Meshing Limitations and Compatibilities)

Load Managers

- When running Fluent under LSF or PBS Professional, it is now possible to use the `-scheduler_ppn=<x>` and/or `-scheduler_gpn=<x>` command line options, in order to set the

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

node processes per cluster node and/or the graphics processing units (GPUs) per cluster node, respectively. (Scheduler Options)

Web Interface

- The Outline View is now referred to as the Settings Tab, whose visibility can now be toggled by clicking the corresponding icon in the upper left hand side of the graphics window.
- By right-clicking the **Graphics** branch in the tree, you can use the **Apply Mirror Planes...** context menu item to display the **Apply Mirror Planes** panel where you can define planes around which you can mirror what is displayed in the graphics window.
- If an item in the tree contains a context menu, you can either right-click to display the menu or you can click the icon that appears when you hover over the item in the tree.
- Additional controls are presented under **View options** in the View Arc. For instance lighting controls that can provide subtle lighting effects such as ambient or specular lighting, as well as distance measurement tools.
- There is an additional view-related tool for saving what you see in the graphics window from time to time. Clicking the icon in the upper right-hand side of the graphics window, you can save the current view.
- Unit specifications are available from the **Units** menu option from the **File** menu. This opens the **Display Units** dialog box where you can view and edit the unit settings for a variety of quantities (see The Main Menu for more information).
- When there are multiple processes that need to be monitored, a badge will indicate the number of processes. Clicking the badge opens a **Progress** panel displaying all ongoing progress indicators and their status.
- You now have the ability to create new surfaces when postprocessing your simulation results for meshes, contours, vectors, pathlines, and particle tracks. In the **Surfaces** field, click the '+' button to access the list of possible surface types to add to the list of potential surfaces for your postprocessing objects.
- You can now configure axes and curve settings for a variety of plots, such as histogram plots, XY plots, cumulative plots, residual plots, and report plots.
- You can start and stop the recording of Python journal files using the **File > Write** menu (see The Main Menu for more information).

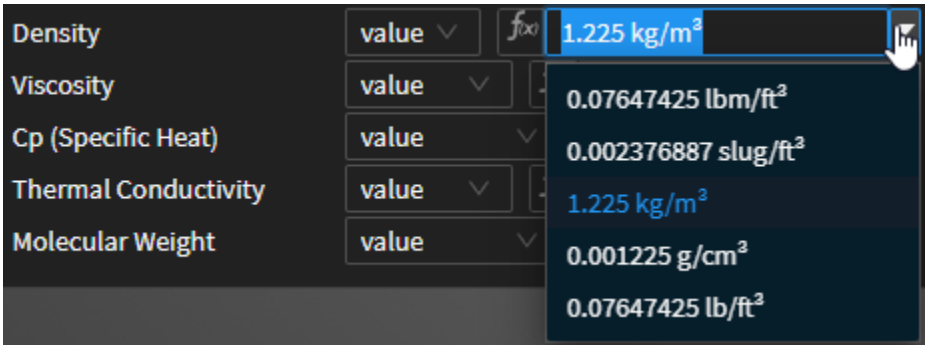
  In addition, you are now able to copy Python commands and paste them into the clipboard, as well as download multiple lines of Python commands to a file on the server (see The Console Panel for more information).
- For panels with tabular data (such as materials-related panels with, for example, polynomial, piecewise polynomial, or piecewise linear data), you can now import or export comma separated values (CSV).

  In addition, for such materials-related panels as well as such panels as the Power Law and Sutherland Law viscosity input panels, you can now preview and plot tabular data.

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

- Several enhancements have been introduced to various panels throughout the web interface, such as:
  - Making relevant commands more readily available, especially in object editor panels.
  - Updating the colormap selection.
  - Improving tabular data inputs (for example, polynomials).
  - Improving in-line searching when selecting field variables.
  - Allowing the height of panels to be adjustable.
- Display is now available at several locations including:
  - Interfaces
  - Cell registers
- The Non-Equilibrium Thermal Model (NETM) is now available in the web interface.
- The mixture multiphase model is now available in the web interface.
- The volume of fluid (VOF) multiphase model is now available in the web interface.
- You can now create new and access existing graphics objects (such as contours, vectors, pathlines, and so on) prior to reading a data file or initializing the flow field.
- Field level unit specification is improved to allow you to type the desired units inline with the value as well as offering a drop-down showing the equivalent value in the other valid units, as shown in Figure 9: Example Inline Unit Drop-down (p. 143).

Figure 9: Example Inline Unit Drop-down



- Mesh Interface creation is improved and simplified, as well as offering diagnostics to help inform you of the overlap which you can use to determine whether the interface is physically relevant. Refer to Accessing Mesh Interface Settings in the *Fluent User's Guide* for additional information.
- You can now review cell zone adjacency and do bulk renaming of cell zones in the **Adjacency** panel, opened from a right-click of the **Cell Zone Conditions** node in the **Settings** tree.

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

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 2026 R1 Beta Features Manual*.

2.3. Fluent Workspace Applications

New features available in the client applications of Ansys Fluent 2026 R1 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.

- **Constant roughness output for iced surfaces**

  The beading roughness output option in the ice physical models section updated to provide a choice between constant roughness or beading as the output roughness height from ICE3D. Constant roughness option will output the user specified roughness height on the iced areas only.

- **Water film reinjection with DPM (beta)**

  Water film that accumulates at trailing edges or separation lines can now be released into the flow with the Lagrangian particle tracking model in the multi-shot and CHT workflows.

- **Particle impingement update during CHT anti-icing with DPM (beta)**

  The anti-icing simulations using Lagrangian tracking and impingement can now have the particle tracks reevaluated at each CHT loop, to recompute collection efficiency with changes in droplet splashing and bouncing and ice crystal sticking efficiency. This capability can also be coupled with DPM water film reinjection to represent water film shedding from heated surfaces.

Changes to existing features and behavior

- **DPM mixing plane**

  Mixing plane implementation that transfers particles collected at the exit of an upstream turbomachinery row to the downstream row is updated for accuracy and robustness. Particles that arrive at the mixing plane outlets are collected in a high-resolution bin distribution with individual flow rate radial profiles. Vapor flow rate profile is included as a separate bin before the particle bins. The `out.mpdat` file that lists these radial profiles for each bin can be used to plot changes in radial flow rate distributions of bins as they move from one row to the next. Effects of physical phenomena like ice crystal shattering (if provided via UDF) can be observed as changes in particle size distribution and flow rates past rotors.

- **ICE3D solution field profile macros for multishot UDFs**

  Some ICE3D solution fields are now automatically extracted as Fluent boundary profile data which can be used as inputs for UDFs in the next shot in multi-shot workflows with UDF

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

macros specifically assigned to these variables. This replaces the need to define UDMs and assign icing values to them at the completion of an ice run, simplifying the use of icing solution in particle/wall interaction UDFs that control splashing, bouncing, shattering, and sticking efficiency.

- **Multishot runs**

  – Multishot runs will no longer remesh the last shot and list the last shot + 1 entry in the project view. Calculations will stop with the final ice step. Users can continue the multishot resuming from the mesh update step of the last shot and increasing the number of shots. This avoids waiting for the last remeshing step when it is not needed.

  – The first shot's mesh will not be loaded automatically when a multishot ends. This is done to keep the last shot's volume solution fields in memory to allow post processing. When using very large grids, having to reload the last shot's case and data files to post-process is costly. Now the last solution will be readily available at the end of the run.

  – To start a new multishot from the original grid, users MUST hit the Reset Multishot button to reload the original grid, as this is no longer done automatically. If there is a restart air solution to be provided, this has to be loaded manually as well after reloading the initial case file.

  – Custom multishot feature corrected, and the button to display the shot-by-shot settings is moved adjacent to the drop down menu. The only available parameter to vary in this version is the icing time per shot.

  – If remeshing fails during a multishot run, the last shot will be reloaded and restart step will be set to Mesh Update. Users can make changes in meshing settings like edge sizing, material point coordinates and wrap resolution factor to retry remeshing.

- **De-icing CHT run memory management**

  A defect with the FENSAP library was causing a cumulative memory leak in the multishot workflows, which resulted in de-icing simulations with many time steps and inner iterations to crash when the available memory is completely filled. The problem is corrected which extends the number of time steps de-icing simulations can run.

- **DPM inlet velocity specification**

  Lagrangian tracking can now assign air flow velocities to particle tracks at inlets. This is done by unchecking the velocity option on the inlet boundary panel for particles, as done similarly for the Eulerian (DROP3D) implementation.

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

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

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.

- **Mesh Adaption**
  - A new combined Hessian Indicator has been introduced that uses Entropy (Combined Hessian - Entropy) instead of Turbulent Viscosity (Combined Hessian - Turbulence). This indicator has been identified as more generic and cost-effective when producing accurate RANS results during the High-Lift Workshop 5.
  - Extra wall monitors have been added to track average temperature and heat flux convergence during adaption cycles as appropriate for high-speed flow simulations.
  - It is now possible to Interrupt an adaption cycle and to directly continue to the next cycle or the next design point without losing the interrupted cycle data.
- **Expansion of Unit Systems (Beta)**
  - An additional unit system has been added to Fluent Aero, US Engineering Common. This new unit system is similar to the US Engineering system but with changes to the units representing angle, length and temperature quantities.
  - The latest unit system used in a simulation is automatically saved and reloaded between simulations.
- **Overset Meshes for Parametric Analysis (Beta)**
  - Geometric Parametric Analyses can be conducted using Overset Meshes in Fluent Aero. In this release, an aircraft component inside an overset mesh can be combined with a background mesh, that contains the aircraft, to conduct parametric simulations by varying the angular position of this component per design point. Therefore, it is now possible to study the impact of an aircraft component on the performance of the entire aircraft.
  - Multiple overset meshes can be supported in Fluent Aero-with or without mesh adaption per design point.
  - Geometric Parametric Analyses with Overset can be conducted using RANS and URANS in 2.5D and 3D simulations in Fluent Aero.
- **Transient Calculations (Beta)**
  - URANS calculations have been introduced in Fluent Aero to capture the natural transient flow behaviour of certain conditions per design point.
  - Convergence Criteria and Default or Robust Convergence Settings based on best practices have not been introduced and therefore these settings should be specified in the baseline case file.
  - Transient calculations can start from a previously steady state solution computed with or without mesh adaption in Fluent Aero.

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

  - Intermediate URANS solutions can be saved to post-process results.
  - Only instantaneous aerodynamic forces and moments are provided through tables and graphs of Fluent Aero. Time-averaged solutions are not provided.
  - Prescribed time-dependent sinusoidal oscillatory angular motions are supported at each overset aircraft component to study transient aerodynamic responses.

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.

- **Polyflow Workspace Add-In for Workbench**

  Gives access to a complete workflow, geometry modelling to results analysis, in the Workbench environment.

- **Statistical analysis**

  Allows statistical analysis as in the Polystat tool of the Polyflow Classic. Relies on CFD-Post to create relevant animations.

- **Adaptive meshing for large variations of field**

  Allows to refine the mesh to limit the variation of specific fields on a finite element.

- **Surface tension for 2D and 3D free surfaces**

  Allows to include surface tension forces in flow modelling and dynamic contact point in 2D.

- **Calibrator model**

  Allows to simulate actual calibrators that shape and cool extrudate as well as conveyors that support profiles in extrusion lines.

- **Mesh quality postprocessor (beta)**

  Allows to display element quality metric when transient mesh deformation occurs, possibly with mesh adaption.

- **Velocity imposed on a free surface (beta)**

  Allows to prescribe the motion of a free surface via its velocity in transient simulations.

- **Electrical heating (beta)**

  Allows to calculate the electric potential field and its joule effect on the fluid temperature.

3. Updates Affecting Ansys Fluent 2026 R1 Code Behavior

The following sections list the code changes in Ansys Fluent 2026 R1.

3.1. Meshing Mode

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

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 2026 R1 that may cause behavior and/or output that is different from the previous release.

Mesh Generation

- When using the Rapid Octree mesher, the following improvements affect the code behavior:

  – The smooth mesh coarsening option is now enabled by default for the Rapid Octree mesher, so that the refined cells follow the geometry better. This option increases cell quality in most cases, and can reduce the overall cell count while maintaining a comparable level of accuracy. If you do not want to take advantage of this option, you can disable it by entering the following text command: `mesh/rapid-octree/mesh-sizing/smooth-mesh-coarsening? no`. (Mesh Parameters)

  – When you specify a reference size (in order to precisely set the edge length for the maximum cell size), the Rapid Octree mesher bounding box will now retain its extents better than in previous releases. Typically it will not be revised at all, and if it is revised, it will only undergo a slight change (up to the maximum cell size). For setups that use the **Bound by Bounding Box** or **Selected Material Points** for the **Volume Specification**, you should still verify that the bounding box is properly located after specifying the reference size. (Defining the Reference Size)

- In previous releases, reading a case file into the meshing mode of Ansys Fluent and then writing out a mesh file would store the original case file version number in the resulting mesh file. If this mesh file was used to set up simulations in later versions of the solver mode of Ansys Fluent, this could result in older solver defaults / behaviors being used unexpectedly.

  In release 2026 R1, this case file version information is omitted from any mesh file written by meshing mode. Note that this change does not affect meshes newly created from geometry in meshing mode, as case file version info was never present unless starting from a case file.

3.2. Solution Mode

This section contains a list of code changes implemented in the solution mode of Ansys Fluent 2026 R1 that may cause behavior and/or results that are different from the previous release.

Solver-Meshing

- The following improvements are now available when fusing face zones:

  – The fusing performance is much faster, especially when fusing a large number of face zones.

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

  - When fusing multiple zones, the naming convention for the resulting zones is improved to be that of the first zone of each fused zone pair, which can help to identify and track the zones.

  - The information printed in the console during fusing is improved: unnecessary warnings are removed, more detailed information is provided (including the IDs of the fused zone pairs and the resulting zones), and a summary is included at the end that provides the number of zone pairs that were fused.

  For information about fusing face zones, see Fusing Selected Face Zones.

- When you perform a mesh check, additional checks are now performed to detect mesh degeneracy. You should repair the mesh if any degeneracies are found, as they may cause the solver to fail or may compromise the solver robustness. (Checking the Mesh)

Solver-Numerics

- Minor inconsistencies are now corrected in the wall distance calculation (when using the default geometric method) for double-precision cases.

Cell Zones and Boundary Conditions

- Fluent now has an improved method for handling name conflicts for cell zones and boundary zones. A name conflict arises when a new zone is created (by actions such as copying a cell zone, appending a case file, and so on) and the name initially generated with the default convention is already used by an existing zone, and so the proposed zone name must be modified. The new handling for name conflicts produces names that have improved consistency and readability, and may be more intuitive for wall-shadow pairs. Note that while the names for new zones may differ compared to those generated in previous releases, the IDs will remain the same.

  For example, one of the changes in handling name conflicts can be seen in the following: if you were to repeatedly append the same case file with a cell zone named `fluid` using the previous release, you would end up with cell zones named `fluid.1`, `fluid.1.1`, and `fluid.1.1.1`, whereas in the current release those same zones would be named `fluid.1`, `fluid.2`, and `fluid.3`, respectively.

Multiphase Models, DDPM Models

- The solution results may now be more accurate for granular multiphase cases that involve the filtered drag model in the Euler/Euler model and the DDPM model on coarse meshes due to a correction of the terminal velocity calculations.

- For cases that use the incompatible combination of the alternative energy treatment for interphase mass transfer and real gas property (RGP) tables, the default sensible enthalpy formulation will be used instead of the alternative formulation. Consequently, you may notice changes in solution results, as the default formulation properly accounts for the heat source due to mass transfer.

Fluent Native GPU Solver

- The default group size for the coupled linear solver is now reduced from 8 to 4, in order to improve the robustness of the coupled linear solver. To revert this change, enter the following

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

Scheme command in the console: (`rpsetvar 'gpuapp/amg-coupled/group-size 4`).

- When using the pressure-based solver with a segregated scheme for the pressure-velocity coupling, the linear solver now respects the cycle type that you have specified for pressure. This is used primarily for some reacting flow workflows where this setting is set explicitly to enhance robustness. To revert this change, set the pressure cycle type to its default setting of V-Cycle (for example, in the **Multigrid** tab of the **Advanced Solution Controls** dialog box).

- At porous interfaces, pressure is now extrapolated in the normal direction only, in order to improve robustness issues at fluid-porous interfaces with skewed meshes. There is no way to revert this improvement.

- For compressible transient flows that use the pressure-based solver with a segregated scheme for the pressure-velocity coupling, the solver now uses the pressure relaxation factor for the compressible transient linearization to give faster convergence within a time step, and for consistency with the CPU solver. To revert this change, enter the following Scheme command in the console: (`gpuapp-set-bool “flow/full-transient-compressible-pressure-relaxation" #t`).

- For LES / SBES simulations, the discretization of normal stress at walls and of the velocity gradients used to compute the turbulent viscosity have been modified to improve robustness. To revert this change, enter each of the following Scheme commands in the console:

  – `(gpuapp-set-string "turbulence/normal-stress-option-at-walls" "turbulent")`

  – `(gpuapp-set-bool "turbulence/use-lsq-grad-for-turbulent-viscosity" #f)`

Graphics, Reporting, and Postprocessing

- The iso-surface and iso-clip algorithm is improved:

  – For iso-surfaces, it offers improved performance, particularly when creating multiple surfaces at once. It also has improved handling of boundaries during surface creation. Note that there may be some differences when comparing with previous releases. In particular, minor changes in appearance may occur when all edges are displayed. Faceting may also be slightly different.

  – For iso-clips, faceting is improved. Note that there may be a noticeable change in the facet count or in edge displays.

  Use the `surface/settings/previous-defaults/pre-26.1-algorithm` text command to revert to the earlier algorithm.

3.3. Fluent Workspace Applications

This section contains a list of code changes implemented in the client applications of Ansys Fluent 2026 R1 that may cause behavior and/or output that is different from the previous release.

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

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 2026 R1, 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.

Graphics, Reporting, and Postprocessing

- The Imprint Surface feature is deprecated and will be removed at a future release. The dialog box is already removed, but the feature is still available using the `surface/create-imprint-surface` TUI command. The recommended replacement is External Surfaces (External Surfaces).

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

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

CFX Release Notes

The following sections contain release information for Release 2026 R1 of Ansys CFX.

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:

- 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 2026 R1 of Ansys CFX.

- There are two new **GUI style** (theme) options: `Dark` and `Light`.

  When starting CFX-Pre (or any other CFX software) for the first time, a dialog box appears that enables you to choose a GUI style (theme). When a GUI style is selected, it is stored as the **GUI Style** user preference for future sessions. For details, see Starting CFX-Pre in the *CFX-Pre User's Guide*.

- Aerodynamic Damping monitors are now more like expression monitors:

  – You can define monitor statistics for aerodynamic damping monitors.

  – Within a CEL expression, you can use the `probe` function to refer to the current value from an aerodynamic damping monitor.

- For Generalized Force monitors in transient blade row cases that have time integration and that do not accumulate Fourier coefficients as part of the pitch-change model, there is a new **Pressure Definition** option: `Pressure Excitation Frequency`. With this option, CFX internally accumulates pressure Fourier coefficients on the specified **Location** only, for the specified **Excitation Frequency**. The real and complex coefficients of the first harmonic order are then used in the Generalized Force calculation. For details, see [Monitor Name]: Pressure Definition in the *CFX-Pre User's Guide*.

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

- You can optionally specify three parameters of the Young growth model for droplet condensation, namely **Young Alpha**, **Young Beta**, and **Condensation Coeff.**, using CEL expressions. For details, see the note at the end of The Droplet Condensation Model in the *CFX-Solver Theory Guide*.

- In CFX-Pre, for convenience when handling a large number of meshes, the **Reload Mesh Files** dialog box now has a **Select All** check box for selecting/clearing all mesh-specific check boxes listed in the dialog box. You could, for example, clear all check boxes before selecting some of them. In addition to the **File** menu command **Reload Mesh Files**, a new shortcut menu command, **Edit Mesh File Paths and Reload**, provides access to the **Reload Mesh Files** dialog box. For details, see Reload Mesh Files Command in the *CFX-Pre User's Guide*.

- The following enhancements were made to the ANSYS CFX-Solver File Editor:

  – A shortcut menu has been added to the object tree. For details, see Shortcut Menu Commands in the *CFX-Solver Manager User's Guide*.

  – The **Edit** menu has a new command, **Options**, to provide access to user preferences. For details, see Options Command in the *CFX-Solver Manager User's Guide*.

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 R2), and the support changes (functionality that is no longer supported) in Release 2026 R1 of Ansys CFX.

- In the user interface, aerodynamic damping monitors are now defined in a slightly different location. In the previous release, aerodynamic damping monitors were listed separately, after the list of point and expression monitors. Starting with Release 2026 R1, aerodynamic damping monitors are listed along with point and expression monitors. To create an aerodynamic damping monitor, you need to select the `Aerodynamic Damping` option under **Monitor Points And Expressions**. For details, see [Monitor Name]: Option in the *CFX-Pre User's Guide*.

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

CFD-Post Release Notes

The following sections contain release information for Release 2026 R1 of Ansys CFD-Post.

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:

- 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 2026 R1 of Ansys CFD-Post.

- There are two new **GUI style** (theme) options: `Dark` and `Light`.

  When starting CFD-Post for the first time, a dialog box appears that enables you to choose a GUI style (theme). When a GUI style is selected, it is stored as the **GUI Style** user preference for future sessions. For details, see Starting CFD-Post in the *CFD-Post User's Guide*.

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

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

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
- BladeGen and BladeEditor
- 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:

- BladeBuilder (see BladeBuilder 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 2026 R1 of TurboSystem.

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

Ansys BladeGen

- NDF export commands have been updated:
  - **File** > **Export** > **Neutral Data File (BladeBuilder)**: This command exports in the newer profile point NDF format (for passing geometry to BladeBuilder).
  - **File** > **Export** > **Neutral Data File (BladeEditor)**: This command exports in the traditional camber-line/thickness NDF format (usually for passing geometry to BladeEditor).

  Note that you can open (using **File** > **Open**) an NDF file from BladeEditor using file filter option "`Neutral Data File (BladeEditor) (*.ndf;*.xml)`".

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

TurboGrid Release Notes

The following sections contain release information for Release 2026 R1 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 2026 R1 of Ansys TurboGrid.

- When TurboGrid runs BladeBuilder to create a CAD model from an NDF file, BladeBuilder shares the TurboGrid license so does not require a BladeBuilder license. When you run BladeBuilder as a standalone application, BladeBuilder still requires a BladeBuilder license.
- When generating CAD from profile points, there is now support for blades with **Cut-off or square** leading or trailing edges, provided that you use BladeBuilder as the **CAD Generation Tool**. For details, see Defining Geometry from Generated CAD: CAD From Profile Points in the *TurboGrid User's Guide*.
- Meshing with high-fidelity geometry now supports splitter blades, provided that the leading and trailing edges are rounded. For details, see Meshing High-Fidelity Geometry in the *TurboGrid User's Guide*.
- There are two new **GUI style** (theme) options: `Dark` and `Light`.

  When starting TurboGrid for the first time, a dialog box appears that enables you to choose a GUI style (theme). When a GUI style is selected, it is stored as the **GUI Style** user preference for future sessions. For details, see Starting Ansys TurboGrid in the *TurboGrid Introduction*.

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

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

BladeBuilder Release Notes

The following sections contain release information for Release 2026 R1 of Ansys BladeBuilder.

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.

2. New Features and Enhancements

This section lists features and enhancements that are new in Release 2026 R1 of Ansys BladeBuilder.

- When TurboGrid runs BladeBuilder to create a CAD model from an NDF file, BladeBuilder shares the TurboGrid license so does not require a BladeBuilder license. When you run BladeBuilder as a standalone application, BladeBuilder still requires a BladeBuilder license.

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

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

Polyflow Classic Release Notes

The following sections contain release information for Ansys Polyflow Classic 2026 R1.

1. New Features
2. Supported Platforms
3. Resolved Issues and Limitations in Ansys Polyflow Classic 2026 R1

1. New Features

The new features in Ansys Polyflow Classic 2026 R1 are as follows:

- It is possible to run a calculation using several MPI threads and multiple GPUS on a cluster. To activate this beta feature, set the environment variable to **`ANSYS_POLYFLOW_BETA_MPI_GPU=1`** (Beta).

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 2026 R1

This section lists limitations that existed in previous releases, but that were removed in Ansys Polyflow Classic 2026 R1.

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

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

Forte Release Notes

The enhancements and defect corrections listed below are for Release 2026 R1.

1. New Features and Enhancements
2. Resolved Issues and Limitations

1. New Features and Enhancements

This section lists enhancements in Release 2026 R1 of Ansys Forte CFD, organized by topic.

Simulation Interface

- **Beta** in 2026 R1:
  - An experimental feature was added to allow users to input a custom Deflection vs. Load profile instead of supplying a fixed elastic modulus for the Beam Bending model using the Built-In FSI.

Job Submision, Monitoring, and Running Options

- Intel MPI 2021.16.1 is the default version.
- Opting In or Opting Out of the Performance Improvement Program is now handled by the Licensing Settings Application and no longer the Forte User Interface.

2. Resolved Issues and Limitations

For a list of Ansys Forte issues that have been resolved in Release 2026 R1, refer to the Resolved Issues and Limitations document on the Ansys Help site.

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

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

Chemkin / Reaction Workbench / MFL Release Notes

The following sections contain release information for Release 2026 R1 of Chemkin products:

1. New Features and Enhancements
2. Resolved Issues and Limitations

1. New Features and Enhancements

Chemkin

- New reactor model is introduced in Release 2026 R1: Flammability Calculator model to compute rich and lean flammability limits.
- pyChemkin:
  - PSR network with heat exchange between the PSRs.
  - ZND, incident-shock, and reflected-shock reactors.
  - Burner-stabilized premixed flame model (given temperature profile and energy equation).
  - Opposed-flow flame models.
  - Multiprocessing/multithreading examples: brute-force sensitivity and parameter study.
  - Advanced application examples: pseudo-stochastic HCCI engine model, SI engine optimization using the pymoo package.
  - Utilities: Partial mixing of two gas mixtures (IEM model), random number generator, normalized species diffusion velocities between two gas mixtures.
- Expanded ability to submit Chemkin jobs to clusters through an offline parameter study capability. Post-processing capability is now available, and the ability to submit reactor network cases is now available (more than one reactor), provided there are no tear loops. See Running and Post-processing from Batch Scripts in the *Chemkin Advanced Analysis Guide*.
- Reactions with more than 259 characters can now be handled.
- More capabilities with Chemkin/GT-Power co-simulation: The Chemkin multi-zone HCCI model can be used as the external cylinder for GT-Power co-simulation.
- Inter-zonal heat transfer is now available to be considered in the Chemkin multizone HCCI model.
- The default connection for Workbench-Chemkin is now changed to a secure connection that requires MTLS certificates.

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

- Opting In or Opting Out of the Performance Improvement Program is now handled by the Licensing Settings Application and no longer the Chemkin User Interface.
- Equilibrium fractions can now be calculated when EGR is specified for the closed homogeneous batch reactor model.

Reaction Workbench

- Mechanisms can now be compared in Reaction Workbench. Parameter studies can be set up to account for a range of operating conditions, and predictions from two mechanisms can be compared graphically for user-selected targets.
- CSP is available as a stand-alone mechanism reduction option.

2. Resolved Issues and Limitations

For a list of issues that have been resolved in Release 2026 R1, refer to the Resolved Issues and Limitations document on the Ansys Help site.

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

FENSAP-ICE Release Notes

The following sections contain release information for Ansys FENSAP-ICE 2026 R1.

1. New Features and Enhancements in Ansys FENSAP-ICE
2. Resolved Issues and Limitations

1. New Features and Enhancements in Ansys FENSAP-ICE

FENSAP-ICE 2026 R1 has no new features or enhancements.

2. Resolved Issues and Limitations

For a list of issues that have been resolved in Release 2026 R1, refer to the Resolved Issues and Limitations document on the Ansys Help website.

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

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

EnSight Release Notes

The following sections contain release information for Ansys EnSight 2026 R1.

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 2026 R1!

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 2026 R1. This list is very short due to significant reallocation of effort towards internal integration of our technologies into other tools. In addition, we have some enhancements that have reached Beta state and can be utilized under the guidelines of a beta feature.

- **Qt 6.8.3 Update**

  EnSight has been updated from Qt5 based UI to a Qt6 based UI. This change was brought on to provide improved security, updated visual elements, and a cleaner UI layout. Some UI elements have improved spacing/layout for more consistent Qt implementation. However, this has not resulted in any major change to the UI, its layout, or operation for the user.

- **gRPC Connection Updates**

  EnSight's utilization of gRPC connection mechanisms has been updated for 2026 R1. EnSight, PyEnSight, EnShell, PyEnShell, and DVS all have several new command line options to allow the user to control how these applications establish connections when using gRPC.

- **Fluent Results data consistency improvements**

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

Many updates to both the Fluent export and import of Fluent native files into EnSight have been made to improve the Fluent-centric handling of data, metadata, and the interpretation of the data into a Fluent-centric manner.

- **AcuSolve Native Reader removed**

  The native AcuSolve format reader has been removed from EnSight. All AcuSolve users are encouraged to utilize the existing and currently suggested EnSight format export capability within AcuSolve.

- **USD Export Enhancements**

  The USD export from EnSight has undergone many improvements, including a completely new transient data structure, smaller GPU footprint, better compatibility with omniverse environments, and camera settings.

- **Keyframe Camera Transformations**

  A new option was added to help handle large view transformations within the keyframe dialog. This 'use short path for transform interpolation' option will improve unwanted camera matrix swapping during large transformations.

- **Internal Library Updates**

  Many 3rd party libraries that are used internally have been updated (e.g. Qt, Python 3.12.11, FFMPEG, Monotype fonts, Boost, OpenSSL, etc) to maintain consistency with other applications and corporate wide requirements.

- **Fluent Material**

  The Surface Property Editor dialog now includes a curated list of Fluent Material textures for visual textures. There are also three controls in the UI section to control the application of those new material textures.

- **Rocky Reader Updates**

  The Rocky reader has gone through updates for changes to the format. It also allows the selection the `.freeflow` extension, which selects the Rocky reader to interpret that file.

2. PyEnSight

The following list highlights the enhancements and changes to PyEnSight in 2026 R1.

- **TBD**

  TBD

3. Ansys Dynamic Reporting (Nexus)

See the 2026 R1 version of the Ansys Dynamic Reporting Release Notes for enhancement and change information.

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

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 2026 R1, refer to the Resolved Issues and Limitations document on the Ansys Help website.

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

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

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-phasefree-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 2026 R1 version.

1.1. SPH Adaptive Sizing (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.

The adaptive sizing is an exclusive FreeFlow feature. The SPH Adaptive Sizing Method's main purpose is to give better performance in certain problems, specially for those which is necessary to have a

better physical evaluation in a portion of the physical domain, instead of analysing the entire domain. In such cases, it is interesting to have refined elements just near the region of interest instead of refining all SPH elements. Therefore, using the SPH Adaptive for creating refined SPH elements, can lower the computation time.

Important:

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

1.2. General New Features (SPH)

- IISPH is now the default SPH solver, and it is no longer Beta.
- Pairwise Potential is now the default surface tension model, and it is no longer Beta.
- A capillary-friction model to enhance adhesion to walls when using the pairwise potential surface-tension model was developed.
- The time periodic support for Fluid Inlets was created.
- The SPH Surface Monitor module, which calculates statistics of SPH other than mass flow, was created.
- The Extend SPH Flowmeters were added to measure the average velocity (or temperature) of the flow over a given surface

1.3. General New Features (Solver SDK)

- Support for global simulation variables was added.
- New post forces hook for the vicinities list hook.
- Support for user-defined point cloud properties was added.

For more details see Rocky for Developers.

1.4. Ansys FreeFlow Coupling with Ansys optiSLang Software Feature

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

- Implemented new single-node optiSLang integration.

For more details see Ansys FreeFlow User Manual.

2. Improvements

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

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

2.1. Performance Improvements

- The performance of point cloud visualization was improved.
- A search cut-off distance was applied to point clouds to improve the performance of nearest-point search.
- The performance of SPH volumetric inlet generation was improved.

2.2. User Experience Improvements

- The SPH parameter was removed from the default data tree to keep the tree cleaner.
- Seed calculation based on the Volumetric Inlet center was automated.

For more details see Ansys FreeFlow User Manual.

2.3. Postprocessing Improvements

- A multi-source Eulerian Process was created to calculate mixed-source statistics (e.g., SPH-DEM volume fraction).
- Dynamic keywords for Plot annotations were implemented.
- The Plane Group user process was implemented.
- Support for the Polyhedron user process on Geometries was created.

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 2026 R1 version.

- **SPH Thermal Buoyancy:** This module implements the Boussinesq approximation to add thermal buoyancy to SPH fluid simulations.
- **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.

  – **Flat plate HTC correlation (Beta):** This module implements the flat-plate correlation for heat transfer coefficient calculation.

  – **SPH Implicit Non-Newtonian (Beta):** Implicit version of non-Newtonian models.

Modules Improvements:

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

- The SPH Freeze Outside Region was promoted from Beta to a regular module.

Modules Removed:

- SPH Disable Wall Corrections

For more details see Module Manual.

4. FreeFlow Scripts

In this section you will find new FreeFlow Scripts in the 2026 R1 version.

Important:

Usability Improvements:

All **PrePost Scripts source code has** been moved to the Ansys Rocky PrePost Scripts & Modules Repository on theAnsys GitHub page.

New Ready-to-use Scripts:

- **Export SPH information (General Scripts):** Like the existing script that exports particle information, this script exports SPH information (such as density, velocity, etc.) for all SPH elements in the computational domain at a chosen time step, saving the data as CSV files, either as one file per time step or a single file with all outputs based on the user's selection.

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.

- *Fast 3D Motion Visualization* beta advanced option was promoted to the default visualization mode for geometries.
- Relaxed Incompressibility constraint for IISPH to reduce pressure fluctuations as a Beta Feature was created.
- An Implicit Viscous Force solver to improve stability for high viscous and turbulent flows was created as a Beta Feature. For the implicit non-Newtonian case, a Beta module was implemented.

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

- The Flat plate HTC correlation module was created as a Beta feature.
- The Buyoancy Force Model module was created as a Beta feature.
- See FreeFlow Modules (p. 177) section to review the Readt-to-use modules beta in this release.

For more details see Ansys FreeFlow User Manual.

6. Resolved Issues and Limitations

For a list of issues that have been resolved in Release 2026 R1, refer to the Resolved Issues and Limitations document on the Ansys Help website.

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

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

Ansys Rocky Release Notes

The following sections contain release information for Ansys Rocky™ particle dynamics simulation software 2026 R1.

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** 2026 R1!

1. New Features

In this section you will find all Rocky new features present in the 2026 R1 version.

1.1. General New Features (DEM)

- The MPI support was added as a Beta Feature.

  The support is only for spherical particle shape, with no support for coupling, no support for external modules or built-in modules in which inter-node reductions are required. The MPI enables Rocky solver to run in distributed memory architetures ("inter-node"). Once with distributed parallel computing support, Rocky users can run simulations in multiple computers (or nodes) connected by a network, such as, InfiniBand. Hence, there will be an improvement on performance scalability for DEM sphere simulations.

For more details see Rocky User Manual.

1.2. General New Features (SPH)

- IISPH is now the default SPH solver, and it is no longer Beta.
- Pairwise Potential is now the default surface tension model, and it is no longer Beta.
- A capillary-friction model to enhance adhesion to walls when using the pairwise potential surface-tension model was developed.
- The time periodic support for Fluid Inlets was created.
- The SPH Surface Monitor module, which calculates statistics of SPH other than mass flow, was created.
- The Extend SPH Flowmeters were added to measure the average velocity (or temperature) of the flow over a given surface

For more details see Rocky User Manual.

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

1.3. General New Features (Rocky SDK)

- Support for global simulation variables was added.
- New post forces hook for the vicinities list hook.
- Support for user-defined point cloud properties was added.

For more details see Rocky for Developers.

1.4. Ansys Rocky Coupling with Ansys Fluent Software Features

See below the new features you can use to perform the Ansys Rocky coupling with Ansys Fluent® software:

- Support for 2-way unresolved CFD-DEM coupling using the Fluent VOF method was implemented.
- Support for 1-way coupling between Fluent GPU and Rocky was implemented.

For more details see Rocky User Manual, and Rocky CFD Manual

1.5. Ansys Rocky Coupling with Ansys optiSLang Software Feature

See below the new features you can use to perform the Ansys Rocky coupling with Ansys optiSLang® software:

- Implemented new single-node optiSLang integration.

For more details see Rocky User Manual.

2. Improvements

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

2.1. PrePost Scripting Improvements

See below the PrePost Scripting improvements that this new version brings:

- A method was created for all inlet classes to add a particle through the PrePost Scripting.

For more details see Rocky for Developers.

2.2. Performance Improvements

- General performance improvements for DEM simulations with high-count, low-PSD spherical particle simulations were implemented.
- The performance of point cloud visualization was improved.
- The performance for the volumetric generation of DEM particles was improved.

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

- A search cut-off distance was applied to point clouds to improve the performance of nearest-point search.
- The performance of SPH volumetric inlet generation was improved.

2.3. User Experience Improvements

- The SPH parameter was removed from the default data tree to keep the tree cleaner.
- Seed calculation based on the Volumetric Inlet center was automated.

For more details see Rocky User Manual.

2.4. Postprocessing Improvements

- A multi-source Eulerian Process was created to calculate mixed-source statistics (e.g., SPH-DEM volume fraction).
- Dynamic keywords for Plot annotations were implemented.
- The Plane Group user process was implemented.
- Support for the Polyhedron user process on Geometries was created.

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 2026 R1 version.

New Ready-to-use Modules:

- **1-Way Free Surface:** This module implements the receding fluid level, considering a given receding velocity; the capillary force model acting on particles in contact with the fluid interface level; the buoyancy force model acting on particles in the fluid modeled by the receding fluid level; the Stokes friction force model acting on particles with the fluid modeled by the receding fluid level.
- **Intermolecular Forces:** This module implements the van der Waals force and the steric repulsive force model as an adhesive contact model. It considers particle-particle interactions.
- **SPH Thermal Buoyancy:** This module implements the Boussinesq approximation to add thermal buoyancy to SPH fluid simulations.
- **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.

  – **Flat plate HTC correlation (Beta):** This module implements the flat-plate correlation for heat transfer coefficient calculation.

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

- **SPH Implicit Non-Newtonian (Beta):** Implicit version of non-Newtonian models.

Modules Improvements:

- The SPH Freeze Outside Region was promoted from Beta to a regular module.

Modules Removed:

- SPH Disable Wall Corrections

For more details see Rocky Module Manual.

4. Rocky Scripts

In this section you will find new Rocky Scripts in the 2026 R1 version.

Important:

Usability Improvements:

All **PrePost Scripts source code has** been moved to the Ansys Rocky PrePost Scripts & Modules Repository on theAnsys GitHub page.

All **Scripts Wizards** *(Material Wizard, Coating Visibility Wizard, Calibration Suite, Material Storage, and Closed Loop Motion)* are now distributed through the APP Catalog on the Ansys Developer Portal.

New Ready-to-use Scripts:

- **Closed Loop Motion (Wizard):** The Closed Loop Motion script automates the creation of replicated geometry motion. It will also generate motion frames with high accuracy. This script uses a table of circles to define the movement's path, and there is also a preview panel to visualize it.
- **Script Mixing Index (General Scripts):** This script calculates the mixing index for multiple materials in a mixer based on the proportions of each material across different cells in the domain. It is useful for quantifying and evaluating the degree of mixing or homogeneity of different materials within the mixer.
- **Export SPH information (General Scripts):** Like the existing script that exports particle information, this script exports SPH information (such as density, velocity, etc.) for all SPH elements in the computational domain at a chosen time step, saving the data as CSV files, either as one file per time step or a single file with all outputs based on the user's selection.

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.

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

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.

- *Fast 3D Motion Visualization* beta advanced option was promoted to the default visualization mode for geometries.
- Relaxed Incompressibility constraint for IISPH to reduce pressure fluctuations as a Beta Feature was created.
- An Implicit Viscous Force solver to improve stability for high viscous and turbulent flows was created as a Beta Feature. For the implicit non-Newtonian case, a Beta module was implemented.
- The Flat plate HTC correlation module was created as a Beta feature.
- The Buyoancy Force Model module was created as a Beta feature.
- Distributed Parallel Computing was added as a Beta feature, improving scalability and enabling more complex simulations.*It is only available for DEM sphere particles.*
- See Rocky Modules (p. 183) section to review the Readt-to-use modules beta in this release.

For more details see Rocky User Manual.

6. Ansys Engineering Copilot within Ansys Rocky (Beta)

When the **Experimental (Beta) Features** checkbox is enabled on the **Options | Preferences** dialog this allows you to use the Ansys Engineering Copilot.

Ansys Engineering Copilot delivers content from Ansys content creators in response to your queries. Content sources that are searched include the Ansys Innovation Space, knowledge articles, forum posts, and **AnsysGPT™**.

Ansys Engineering Copilot opens a dock widget in Rocky software, where you can access several resources that will help you find answers to your questions and learn something new by engaging with the large global Ansys community. For more information, see the Ansys Engineering Copilot documentation.

For more details see Rocky User Manual.

Important:

Note that beta features have not been fully tested and validated. Ansys, Inc. makes no commitment to resolve defects reported against these prototype features. However, your feedback will help us improve the overall quality of the product. We will not guarantee that

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

the projects using this beta feature will run successfully when the feature is finally released so you may, therefore, need to modify the projects.

7. Resolved Issues and Limitations

For a list of issues that have been resolved in Release 2026 R1, refer to the Resolved Issues and Limitations document on the Ansys Help website.

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

Thermal Desktop 2026 R1 Release Notes

1. New features and capabilities

The following changes have been added.

General

- TD Direct will be discontinued in a future release and has been replaced by TD Designer in 2026 R1.
- Thermal Desktop Registers can now be used as System Coupling Parameters. This is a beta feature.
- Applications not supported using AutoCAD 2025:
  - ModelCenter integration
  - OptiSLang integration
  - System Coupling client
  - Any user-created OpenTD tools that connect to Thermal Desktop (reading save files is unaffected by AutoCAD version)

  Note:

  Please see the Thermal Desktop manual (Overrides) to enable features if using AutoCAD 2025 or later.

- Modeling of participating media is radiation has been promoted out of Beta. Participating media is a volume of solid, liquid, or gas that emits and absorbs radiation. The volumes may be analyzed for heating rates from external sources and participate in volume-to-volume and surface-to-volume radiation exchange. Only grey media without scattering is supported at this time. Common applications of participating media include the analysis of ports used in manned spacecraft, protective covers for optical systems and solar heat exchangers, lamp design, the analysis of canopies and windscreens for cabin comfort in airplanes and automobiles, and survival analysis for irradiated critical components.
- In 26R1, the compartment evaluation workflow has been optimized to reduce unnecessary geometric computations during wet/dry tracking and interface updates. The solver now avoids redundant recal-

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

culations when the liquid plane is unchanged and skips expensive volume slicing when compartments are fully empty or fully filled.

Note:

Expected results: Fluid models containing compartments may see reduced run time. Performance gains are most pronounced in compartment-dominated cases. No changes to model setup are required, and solution results are unchanged.

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

Ansys Electronics Products

Release notes are available for the following Ansys Electronics products:

Icepak (p. 191)

Icepak Release Notes

Release 2026 R1 of the Ansys Icepak application offers most of the capabilities from previous releases plus an enhancement.

- Introduction (p. 191)
- New and Modified Features in Ansys Icepak 2026 R1 (p. 191)
- Resolved Issues and Limitations in Ansys Icepak 2026 R1 (p. 191)

1. Introduction

Ansys Icepak 2026 R1 is a release of Ansys Icepak that has a new feature and resolved issues and limitations.

2. New and Modified Features in Ansys Icepak 2026 R1

Release 2026 R1 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 2026 R1

For a list of issues that have been resolved in Release 2026 R1, refer to the Resolved Issues and Limitations document on the Ansys Help website.

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

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

Ansys Optical Products

Release notes are available for the following Ansys optical products:

Ansys SPEOS (p. 195)
SPEOS for NX (p. 205)
SPEOS for Creo Parametric (p. 213)
Ansys Optical Labs (p. 219)
SPEOS HPC (p. 223)

Speos Release Notes

The following sections present the new features, enhancements and changes for Speos 2026 R1 release.

1. Features from BETA to Official
2. Productivity Enhancement
3. Sensor / Autonomous Driving
4. Result Experience
5. Optical Part Design
6. Changes
7. Resolved Issues and Limitations

1. Features from BETA to Official

This section highlights the features that have evolved from beta version into fully supported, official features.

Interactive Live Preview

The Interactive Live Preview tool is now official and no longer in BETA.

The Interactive Live Preview tool permits you to directly see in the current running Live Preview window the changes applied on your project without having to launch a simulation to see the result.

2. Productivity Enhancement

This section highlights the new capabilities introduced in Ansys Speos 2026 R1 designed to enhance productivity.

Speos RPC

Refer to the Developer Documentation portal to see the latest changes for Speos RPC.

Speos client APIs

Refer to the Developer Documentation portal to see the latest changes for Speos client APIs.

Ambient Sources

The Ambient source sampling algorithm in Direct simulation has been aligned with the other sources. Consequently:

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

- Ambient sources will have fewer rays for simulations with multiple sources. The Standard Deviation is expected to increase marginally.
- Direct simulations time may be improved, especially when multiple sources and ambient sources are used, and a significant number of rays is set.
- Power Report in the HTML Report now integrates the emitted power information of Ambient sources (see below)
- Results may be slightly noisier as less rays are sent during the simulation compared to versions prior to 2026 R1.



Simulation - Ray animation

In some cases, full ray tracing contains too much information to understand how light propagates and interacts with the different optical interfaces.

- The new ray length parameter, available with Interactive simulations and Light Expert analysis, permits to crop the ray paths and ease the analysis.



- In addition, the **Play function** (Interactive simulations / Light Expert analysis) literally animates the ray tracing, offering the possibility to follow the propagation with different display modes.



Simulation - new 3D Irradiance sensor result

Direct simulation with 3D irradiance sensor now generates a new result that contains all useful information for post processing. This is quite useful, especially when performing a Sun burn analysis.

Information for each 3D face includes:

- Every related to the meshing: facets, vertices and normals
- Radiometric values: irradiance, reflection, transmission, absorption, total power and absorbed power
- Data to advanced filtering: area and number of impacts per facet

This new result *.rawxm3 file uses the HDF5 format that can be opened with open-source software or via Python script.



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

Simulation - Contextual menu actions for multi-selection of simulations

You can now perform the following contextual menu actions in case of a multi-selection of simulations. The action selected will be performed for each selected simulation:

- GPU Compute
- Isolate
- Linked Export
- Export

This will streamline your workflows by eliminating the need to apply actions to each simulation individually, therefore saving time and improving efficiency.



Simulation - Creating a simulation based on an existing simulation

Speos now offers the possibility to create a new simulation from an existing one (the simulation type can be different) in few clicks. Compatible features used in the selections of the original simulation are transferred to the new one, either all at once (sources, sensors and geometries) or individually. It eases the setting up of new simulations. For instance, by reusing the selections from an Interactive simulation to a Direct or Inverse simulation.



Live Preview

In the **Simulation display options** of the Live Preview, a new Preview mode has been added to define how Live Preview displays results during static and dynamic view states to provide different user experiences.

- The new **Hybrid mode** has been designed for visual fluidity during result navigation:
  - When the viewpoint is static, Live Preview uses the GPU rendering. The Live Preview simulation continues running.
  - When moving the viewpoint, Live Preview switches to a traditional CAD colored render to offer better motion fluidity. During the movement, Live Preview simulation is paused and resumes as soon as the movement stops and immediately displays back simulation rendering.
- The **GPU mode** corresponds to the original mode, and prioritizes GPU simulation visual over motion fluidity.
  - When the viewpoint is static, Live Preview uses the GPU rendering. The Live Preview simulation continues running.

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

– When moving the viewpoint, Live Preview keeps running the simulation but at a much lower resolution to improve interactivity. Resolution gradually increases back to full resolution as soon as movement stops.



Ansys Cloud Burst Compute

Ansys Cloud Burst Compute has been improved with the following new functionalities:

- The new **Disable ray files and lpf/lp3 files output** option is available in the job submission window during the creation of the simulation jobs. As these files can take time to be generated, disabling this option speeds up the simulation generation if you do not need them.
- The new **Download results after completion** option is available in the job submission window. Thus, when activating this option, the simulation results are automatically downloaded when the job is in **Finished** status.
- The new **Stop and save** option is available in the Ansys Cloud Burst Compute Job Manager window (job monitoring window). This option appears only when a job is running. It stops the simulation job and saves all results up to the current point.



Ansys Engineering Copilot

Ansys Engineering Copilot is now available to every user, expanding what you can do right away. AnsysGPT stays part of the enhanced experience offered with an active **AnsysGPT AI+** license.



Speos Core

Speos Core has been improved to provide you with a better user experience:

- Speos Core has been optimized to provide better memory management when running multiple simulations in a row.
- Speos Core offers the possibility to set up GPU preferences as you will do in Speos through the new **GPU solver configuration** button available in the SPEOS Core toolbar.
- Speos Core gives you access to the Simulation folder through the new **Simulation folder** button available in the SPEOS Core toolbar.
- After a compute, simulation results and report can be opened from the Speos Core tree. This direct access saves time to find the corresponding files on the hard disk drive.



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

Libraries

As Ansys Optics Library is available in the installation package, Speos now offers a direct access to it through:

- Material Libraries: all optical properties references are loaded and ready to be used in project with simple drag and drop.
- Libraries: shows all the Optics library content (optical properties, sensors, sources and standards)

In addition, a direct access to List file references tool is possible from the ribbon. The tool permits to easily manage paths to dependencies as well as related files.



3. Sensor / Autonomous Driving

This section highlights the new capabilities introduced in Ansys Speos 2026 R1 designed to enhance sensors and autonomous driving.

Optical Design Exchange

- The Optical Design Exchange construction algorithm has been improved to deliver greater robustness when importing a *.odx file and computing the Optical Design Exchange feature in Speos.
- Optical Design Exchange now supports Table Glass and creates at the import the corresponding *.ZTG.ZemaxMaterial file.

  Table Glasses in Ansys Zemax OpticStudio® are optical materials in which the material properties are defined in a text file (*.ztg) explicitly, defining, for each wavelength, index of refraction, transmissivity and corresponding thickness.

  When exporting a *.odx file to Speos, Table Glass data are carried in an encrypted *.ZTG.ZemaxMaterial file that contains:

  – Wavelengths in nm

  – Dispersion coefficients (refractive index) for each wavelength

  – Absorption coefficients in mm-1, calculated from the transmission and thickness data

Camera Sensor - XMP Template

Camera sensor offers the possibility to reference a XMP template (*.xml file) during its definition.

- The template contains useful information such as default measurements that will be automatically applied to new spectral irradiance or exposure result.

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

- In addition, you can choose to setup dimensions of the camera sensor from the template and to apply display settings to new results.

XMP template contributes to save time during simulation iterations using camera sensor.



4. Result Experience

This section highlights the new capabilities introduced in Ansys Speos 2026 R1 designed to enhance the result experience.

CPU/GPU solver algorithms

- CPU and GPU solver algorithms have been aligned for a better management of **unpolished surfaces**.
- GPU solver algorithm has been optimized to provide better **memory management** when running **multiple GPU simulations** in a row.
- The GPU solver algorithm has been enhanced to ensure more robust execution when running an Inverse simulation with the **Optimized Propagation** set to **Relative**.
- The GPU solver algorithm has been improved to correctly take into account rays intersecting **Exit Geometries** (from Ray File or Surface source) during a simulation.
- The GPU solver algorithm has been aligned with the CPU solver algorithm for a better management of **Display sources** in simulation. Now **Fast Transmission Gathering** algorithm (when activated) automatically applies to the geometry (automatically created by the simulation) of a Display source. Thus, the geometry is ignored by the simulation.

Simulation result - Improved visualization of results using transparency

Speos now permits to change the transparency of results displayed in 3D view, enhancing the visibility of underlying elements, and enabling more effective analyses such as ray tracing or Light Expert. In addition, xm3 results null values can be displayed as transparent. These new options are available in the **Light Simulation** options of the SpaceClaim Options.



Simulation result - HTML GPU Report

Now, GPU simulation report provides the same level of information as CPU simulation.

- **Propagation report** section has been added and includes the error report that provides the total number of errors and the list of error types.
- In addition, the **Performance Analysis** section of the report includes peak video memory usage that corresponds to the highest amount of vRAM consumed by the simulation at a certain time.

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

This information becomes quite useful when you want to estimate how much the resolution of sensors can be increased for instance.



GPU acceleration - Rebalancing Camera simulation

Camera sensors in GPU Inverse simulations are now compatible with:

- the Optimized Propagation set to **Relative** when running the simulation.
- the Rebalancing mode set to **Variance** when running the Live Preview.

Virtual Photometric Lab / Virtual Human Vision Lab

- As of version 2026 R1, XMP files generated are in **version 10.3**. XMP files generated prior to version 2026 R1 will not load in the version 2026 R1 of Speos, Virtual Photometric Lab, and Virtual Human Vision Lab.
- XMP Filtering is now deactivated by default (set to None) for XMP results generated with a sensor without XML template applied. Consequently, simulations will require less memory during result generation and will deliver faster result output, enabling Virtual Photometric Lab / Virtual Human Vision Lab to open the results more quickly.



5. Optical Part Design

This section highlights the new capabilities introduced in Ansys Speos 2026 R1 designed to enhance Optical Part Design features.

Optical Part Design - Updating Multiple Parameter Values Simultaneously

To simplify parameters definition, Speos introduces the possibility to edit multiple rows of a parameter in a table by using the new multi-select column editing feature. Thus, you can select several cells within a single parameter column and immediately apply a new value to all of them at once, which will save you time in feature setting. This new feature is mainly beneficial for Optical Part Design features but can be used with other tables of Speos features if allowed.



Optical Part Design - Optical Surface / Optical Lens

With the **Custom Contour** tool, it is now possible to use multiple surfaces to trim the Optical Surface / Optical Lens. Usually, those multiple surfaces come from style, mechanical, multiple parts assembly. Prior to version 2026 R1, only a surface body was allowed to perform the trimming action.

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



Optical Part Design - TIR Lens

Total Internal Reflection Lens can now be defined using an optical axis that can be tilted with respect to the support plan where LED is placed.

- Prior to version 2026 R1, TIR lens design was based on a support plane, a defined source point, and an automatically generated axis of revolution (the optical axis) normal to the plane. This approach inherently produced symmetrical lenses.
- Since version 2026 R1, the Optical axis has become a user-defined parameter. TIR Lens design continues to rely on a support plane, a source point, and the Optical axis itself. However, the Optical axis is no longer constrained to be normal to the support plane. It can now be oblique, allowing for the creation of asymmetrical lenses.

This can be used also inside Optical Lens as entry face with tilted PCB to reduce space required by the optical system.



Optical Part Design - Light Guide

- The Light Guide construction algorithm has been enhanced to provide more robust results.
- Hybrid mode of Light Guide is extremely flexible as prism could be inside or outside main body. To bring more flexibility, The Max prism height manufacturing parameter has been added to the Light Guide definition. It allows you to limit the height of each prism from the guide curve, preventing any collision with a mechanical element. Thus, you can optimize your light guide while considering various manufacturing and assembly constraints.



Optical Part Design - Freeform Lens

Speos introduces a new type of Freeform Lens dedicated to create a sharp cut-off lens from a multichip LED. Such a lens is particularly efficient for road lighting (automotive or motorbike).

Thus, to create such a lens, new parameters have been introduced to define the Back Face of the Lens and the Target Beam rendering on the result. Upon compute of the Optical Part Design feature, the geometric design is automatically optimized to achieve the user-defined optical target.



HUD Optical Design

HUD Optical Design now generates the rotation axis and gives access to the corresponding angles when defining more than one eyebox and enabling multiconfiguration.

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



HUD Optical Analysis

HUD Optical Analysis algorithm has been enhanced to better detect stigmatic systems and inform the user of the situation where Sagittal Virtual Image, Tangential Virtual Image, and Best Focus Virtual Image are coincident.

6. Changes

This section describes the operational and support changes introduced in Ansys Speos 2026 R1 release.

Ansys Product Improvement Program

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

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

Documentation

- Documentation has been updated with a new chapter explaining how to access and collect logs. This addition makes troubleshooting, debugging, and other diagnostic tasks easier. Besides, it also provides the development team with possible useful information to investigate and resolve software issues more efficiently.
- To improve clarity and provide a more coherent workflow, the Simulations and Results documentation chapters have been reorganized and refined.
  - **Restructured Simulation chapter**: The Simulation content is now organized into clearer, stage-based sub-sections that better reflect the progression of the simulation process.
  - **Results integrated into the workflow**: The former Results chapter has been moved under Simulation and renamed Analyzing Results, placing outcome information directly alongside the steps that produce it.
  - **Enhanced navigation**: The table of contents has been updated to reflect this new structure, making it easier for users to follow the end-to-end process.

Deprecated / End of support / End of life features

- **SDOC** file format is in **end of support in version 2026 R1**, meaning that *.scdoc files cannot be opened from version 2026 R1. If you want to open a SCDOC project, convert the *.scdoc file to *.sdocx by saving it from a prior version (2025 R1 or 2025 R2 for example), then open the *.scdocx file in 2026 R1.

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

- **FTM Layer** is deprecated and in **end of support in version 2026 R1** and will be in **end of life in version 2027 R1**.
- **Solstis ray files** are deprecated and in **end of support in version 2026 R1** and will be in **end of life in version 2027 R1**.
- The following filtering options of Virtual Photometric Lab are deprecated and in **end of support in version 2026 R1**:

- The following filtering options of Virtual Human Vision Lab are deprecated and in **end of support in version 2026 R1**:

- The human vision Dynamic Adaptation option is deprecated and in end of support in version 2026 R1 in the following tools:
  - Virtual Human Vision Lab
  - Virtual Reality Lab
  - Virtual Lighting Animation Tool

  The default **Adaptation type** value is now set to **Dynamic Adaptation 2019**.
- The **Deterministic algorithm** with Photon Map (Build Photon Map, Load Photon Map, Build and Save Photon Map) is in **end of support in version 2026 R1**.
- **Ansys Cloud Direct** is **deprecated on December 31, 2025**. Since this date, job submission to Ansys Cloud Direct are no longer supported. Also, Ansys Cloud Direct documentation is removed from the Ansys Help site in January.

Versions Update

- Windows: Windows 10 and Windows 11

  Minimum NVIDIA Driver **581.42** is required from 2026 R1.
- Speos GPU solver algorithm is now compatible with NVIDIA Blackwell GPU architecture.

7. Resolved Issues and Limitations

For a list of issues that have been resolved in Release 2026 R1, refer to the Resolved Issues and Limitations document on the Ansys Help website.

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

Speos for NX Release Notes

The following sections present the new features, enhancements and changes for Speos for NX 2026 R1 release.

1. Productivity Enhancement
2. Sensor / Autonomous Driving
3. Result Experience
4. Optical Part Design
5. Changes
6. Resolved Issues and Limitations

1. Productivity Enhancement

This section highlights the new capabilities introduced in Ansys Speos for NX 2026 R1 designed to enhance productivity.

Speos RPC

Refer to the Developer Documentation portal to see the latest changes for Speos RPC.

Ambient Sources

The Ambient source sampling algorithm in Direct simulation has been aligned with the other sources. Consequently:

- Ambient sources will have fewer rays for simulations with multiple sources. The Standard Deviation is expected to increase marginally.
- Direct simulations time may be improved, especially when multiple sources and ambient sources are used, and a significant number of rays is set.
- Power Report in the HTML Report now integrates the emitted power information of Ambient sources (see below)
- Results may be slightly noisier as less rays are sent during the simulation compared to versions prior to 2026 R1.



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

Simulation - new 3D Irradiance sensor result

Direct simulation with 3D irradiance sensor now generates a new result that contains all useful information for post processing. This is quite useful, especially when performing a Sun burn analysis.

Information for each 3D face includes:

- Every related to the meshing: facets, vertices and normals
- Radiometric values: irradiance, reflection, transmission, absorption, total power and absorbed power
- Data to advanced filtering: area and number of impacts per facet

This new result *.rawxm3 file uses the HDF5 format that can be opened with open-source software or via Python script.



Live Preview

In the **Simulation display options** of the Live Preview, a new Preview mode has been added to define how Live Preview displays results during static and dynamic view states to provide different user experiences.

- The new **Hybrid mode** has been designed for visual fluidity during result navigation:
  - When the viewpoint is static, Live Preview uses the GPU rendering. The Live Preview simulation continues running.
  - When moving the viewpoint, Live Preview switches to a traditional CAD colored render to offer better motion fluidity. During the movement, Live Preview simulation is paused and resumes as soon as the movement stops and immediately displays back simulation rendering.
- The **GPU mode** corresponds to the original mode, and prioritizes GPU simulation visual over motion fluidity.
  - When the viewpoint is static, Live Preview uses the GPU rendering. The Live Preview simulation continues running.
  - When moving the viewpoint, Live Preview keeps running the simulation but at a much lower resolution to improve interactivity. Resolution gradually increases back to full resolution as soon as movement stops.



Ansys Cloud Burst Compute

Ansys Cloud Burst Compute has been improved with the following new functionalities:

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

- The new **Disable ray files and lpf/lp3 files output** option is available in the job submission window during the creation of the simulation jobs. As these files can take time to be generated, disabling this option speeds up the simulation generation if you do not need them.
- The new **Download results after completion** option is available in the job submission window. Thus, when activating this option, the simulation results are automatically downloaded when the job is in **Finished** status.
- The new **Stop and save** option is available in the Ansys Cloud Burst Compute Job Manager window (job monitoring window). This option appears only when a job is running. It stops the simulation job and saves all results up to the current point.



Ansys Engineering Copilot

Ansys Engineering Copilot is now available to every user, expanding what you can do right away. AnsysGPT stays part of the enhanced experience offered with an active **AnsysGPT AI+** license.



Speos Core

Speos Core has been improved to provide you with a better user experience:

- Speos Core has been optimized to provide better memory management when running multiple simulations in a row.
- Speos Core offers the possibility to set up GPU preferences as you will do in Speos for NX through the new **GPU solver configuration** button available in the SPEOS Core toolbar.
- Speos Core gives you access to the Simulation folder through the new **Simulation folder** button available in the SPEOS Core toolbar.
- After a compute, simulation results and report can be opened from the Speos Core tree. This direct access saves time to find the corresponding files on the hard disk drive.



Libraries

Speos for NX now offers the possibility to customize the path of the Optical Libraries folder (by default in C:\Program Files\ANSYS Inc\vXXX\Optical Products\OpticalLibraries).

In addition, a direct access to Libraries shows all the Optics library content (optical properties, sensors, sources and standards).



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

2. Sensor / Autonomous Driving

This section highlights the new capabilities introduced in Ansys Speos for NX 2026 R1 designed to enhance sensors and autonomous driving.

Optical Design Exchange

Optical Design Exchange is now available in Speos for NX. The Optics Design Exchange feature allows you to import an optical design system created in Ansys Zemax OpticStudio. Each geometry imported (source, lens, mirror, stop) is located into a dedicated part that includes:

- Default properties: volume and surface optical properties, meshing properties
- Specific properties for coatings
- Surface sources for sources

Through the Speos for NX capabilities you will be able to use and analyze this optical design system.



3. Result Experience

This section highlights the new capabilities introduced in Ansys Speos for NX 2026 R1 designed to enhance the result experience.

CPU/GPU solver algorithms

- CPU and GPU solver algorithms have been aligned for a better management of **unpolished surfaces**.
- GPU solver algorithm has been optimized to provide better **memory management** when running **multiple GPU simulations** in a row.
- The GPU solver algorithm has been enhanced to ensure more robust execution when running an Inverse simulation with the **Optimized Propagation** set to **Relative**.
- The GPU solver algorithm has been improved to correctly take into account rays intersecting **Exit Geometries** (from Ray File or Surface source) during a simulation.
- The GPU solver algorithm has been aligned with the CPU solver algorithm for a better management of **Display sources** in simulation. Now **Fast Transmission Gathering** algorithm (when activated) automatically applies to the geometry (automatically created by the simulation) of a Display source. Thus, the geometry is ignored by the simulation.

Simulation result - HTML GPU Report

- Material names in the HTML Report have been renamed from **Vop** to **Optic**.
- Now, GPU simulation report provides the same level of information as CPU simulation.

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

- **Propagation report** section has been added and includes the error report that provides the total number of errors and the list of error types.
- In addition, the **Performance Analysis** section of the report includes peak video memory usage that corresponds to the highest amount of vRAM consumed by the simulation at a certain time. This information becomes quite useful when you want to estimate how much the resolution of sensors can be increased for instance.



GPU acceleration - Rebalancing Camera simulation

Camera sensors in GPU Inverse simulations are now compatible with:

- the Optimized Propagation set to **Relative** when running the simulation.
- the Rebalancing mode set to **Variance** when running the Live Preview.

Virtual Photometric Lab / Virtual Human Vision Lab

- As of version 2026 R1, XMP files generated are in **version 10.3**. XMP files generated prior to version 2026 R1 will not load in the version 2026 R1 of Speos, Virtual Photometric Lab, and Virtual Human Vision Lab.
- XMP Filtering is now deactivated by default (set to None) for XMP results generated with a sensor without XML template applied. Consequently, simulations will require less memory during result generation and will deliver faster result output, enabling Virtual Photometric Lab / Virtual Human Vision Lab to open the results more quickly.



4. Optical Part Design

This section highlights the new capabilities introduced in Ansys Speos 2026 R1 designed to enhance Optical Part Design features.

Optical Part Design - Light Guide

The Light Guide construction algorithm has been enhanced to provide more robust results.

HUD Optical Analysis

HUD Optical Analysis algorithm has been enhanced to better detect stigmatic systems and inform the user of the situation where Sagittal Virtual Image, Tangential Virtual Image, and Best Focus Virtual Image are coincident.

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

5. Changes

This section describes the operational and support changes introduced in Ansys Speos for NX 2026 R1 release.

Ansys Product Improvement Program

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

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

Documentation

- Documentation has been updated with a new chapter explaining how to access and collect logs. This addition makes troubleshooting, debugging, and other diagnostic tasks easier. Besides, it also provides the development team with possible useful information to investigate and resolve software issues more efficiently.
- To improve clarity and provide a more coherent workflow, the Simulations and Results documentation chapters have been reorganized and refined.
  - **Restructured Simulation chapter**: The Simulation content is now organized into clearer, stage-based sub-sections that better reflect the progression of the simulation process.
  - **Results integrated into the workflow**: The former Results chapter has been moved under Simulation and renamed Analyzing Results, placing outcome information directly alongside the steps that produce it.
  - **Enhanced navigation**: The table of contents has been updated to reflect this new structure, making it easier for users to follow the end-to-end process.

Deprecated / End of support / End of life features

- **FTM Layer** is deprecated and in **end of support in version 2026 R1** and will be in **end of life in version 2027 R1**.
- **Solstis ray files** are deprecated and in **end of support in version 2026 R1** and will be in **end of life in version 2027 R1**.
- The following filtering options of Virtual Photometric Lab are deprecated and in **end of support in version 2026 R1**:



- The following filtering options of Virtual Human Vision Lab are deprecated and in **end of support in version 2026 R1**:

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



- The human vision Dynamic Adaptation option is deprecated and in end of support in version 2026 R1 in the following tools:

  – Virtual Human Vision Lab

  – Virtual Reality Lab

  – Virtual Lighting Animation Tool

  The default **Adaptation type** value is now set to **Dynamic Adaptation 2019**.

- The **Deterministic algorithm** with Photon Map (Build Photon Map, Load Photon Map, Build and Save Photon Map) is in **end of support in version 2026 R1**.

- The **Camera Post Processing** is in **end of support in version 2026 R1**.

- **Ansys Cloud Direct** is **deprecated on December 31, 2025**. Since this date, job submission to Ansys Cloud Direct are no longer supported. Also, Ansys Cloud Direct documentation is removed from the Ansys Help site in January.

Versions Update

- Windows: Windows 10 and Windows 11

  Minimum NVIDIA Driver **581.42** is required from 2026 R1.

- Speos GPU solver algorithm is now compatible with NVIDIA Blackwell GPU architecture.

Operating System

- NX 2312 is no longer supported as of 2026 R1.

- NX 2406 up to NX 2406 (2406.9160) service pack is now supported as of 2026 R1.

- NX 2412 up to NX 2412 (2412.8701) service pack is now supported as of 2026 R1.

- NX 2506 is no longer in BETA and now officially released as of 2026 R1.

  NX 2506 up to NX 2506 (2506.6002) service pack is now supported as of 2026 R1.

- NX 2512 up to NX 2512 (2512.1700) service pack is now supported as of 2026 R1.

6. Resolved Issues and Limitations

For a list of issues that have been resolved in Release 2026 R1, refer to the Resolved Issues and Limitations document on the Ansys Help website.

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

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

Speos for Creo Parametric Release Notes

The following sections present the new features, enhancements and changes for Speos for Creo Parametric 2026 R1 release.

1. Productivity Enhancement
2. Result Experience
3. Changes
4. Resolved Issues and Limitations

1. Productivity Enhancement

This section highlights the new capabilities introduced in Ansys Speos for Creo Parametric 2026 R1 designed to enhance productivity.

Speos RPC

Refer to the Developer Documentation portal to see the latest changes for Speos RPC.

Ambient Sources

The Ambient source sampling algorithm in Direct simulation has been aligned with the other sources. Consequently:

- Ambient sources will have fewer rays for simulations with multiple sources. The Standard Deviation is expected to increase marginally.
- Direct simulations time may be improved, especially when multiple sources and ambient sources are used, and a significant number of rays is set.
- Power Report in the HTML Report now integrates the emitted power information of Ambient sources (see below)
- Results may be slightly noisier as less rays are sent during the simulation compared to versions prior to 2026 R1.



Simulation - new 3D Irradiance sensor result

Direct simulation with 3D irradiance sensor now generates a new result that contains all useful information for post processing. This is quite useful, especially when performing a Sun burn analysis.

Information for each 3D face includes:

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

- Every related to the meshing: facets, vertices and normals
- Radiometric values: irradiance, reflection, transmission, absorption, total power and absorbed power
- Data to advanced filtering: area and number of impacts per facet

This new result *.rawxm3 file uses the HDF5 format that can be opened with open-source software or via Python script.



Live Preview

In the **Simulation display options** of the Live Preview, a new Preview mode has been added to define how Live Preview displays results during static and dynamic view states to provide different user experiences.

- The new **Hybrid mode** has been designed for visual fluidity during result navigation:
  - When the viewpoint is static, Live Preview uses the GPU rendering. The Live Preview simulation continues running.
  - When moving the viewpoint, Live Preview switches to a traditional CAD colored render to offer better motion fluidity. During the movement, Live Preview simulation is paused and resumes as soon as the movement stops and immediately displays back simulation rendering.
- The **GPU mode** corresponds to the original mode, and prioritizes GPU simulation visual over motion fluidity.
  - When the viewpoint is static, Live Preview uses the GPU rendering. The Live Preview simulation continues running.
  - When moving the viewpoint, Live Preview keeps running the simulation but at a much lower resolution to improve interactivity. Resolution gradually increases back to full resolution as soon as movement stops.



Ansys Cloud Burst Compute

Ansys Cloud Burst Compute has been improved with the following new functionalities:

- The new **Disable ray files and lpf/lp3 files output** option is available in the job submission window during the creation of the simulation jobs. As these files can take time to be generated, disabling this option speeds up the simulation generation if you do not need them.
- The new **Download results after completion** option is available in the job submission window. Thus, when activating this option, the simulation results are automatically downloaded when the job is in **Finished** status.

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

- The new **Stop and save** option is available in the Ansys Cloud Burst Compute Job Manager window (job monitoring window). This option appears only when a job is running. It stops the simulation job and saves all results up to the current point.



Ansys Engineering Copilot

Ansys Engineering Copilot is now available to every user, expanding what you can do right away. AnsysGPT stays part of the enhanced experience offered with an active AnsysGPT AI+ license.



Speos Core

Speos Core has been improved to provide you with a better user experience:

- Speos Core has been optimized to provide better memory management when running multiple simulations in a row.
- Speos Core offers the possibility to set up GPU preferences as you will do in Speos for NX through the new **GPU solver configuration** button available in the SPEOS Core toolbar.
- Speos Core gives you access to the Simulation folder through the new **Simulation folder** button available in the SPEOS Core toolbar.
- After a compute, simulation results and report can be opened from the Speos Core tree. This direct access saves time to find the corresponding files on the hard disk drive.



Libraries

Speos for Creo Parametric now offers the possibility to customize the path of the Optical Libraries folder (by default in C:\Program Files\ANSYS Inc\vXXX\Optical Products\OpticalLibraries).

In addition, a direct access to Libraries shows all the Optics library content (optical properties, sensors, sources and standards).



2. Result Experience

This section highlights the new capabilities introduced in Ansys Speos for Creo Parametric 2026 R1 designed to enhance the result experience.

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

CPU/GPU solver algorithms

- CPU and GPU solver algorithms have been aligned for a better management of **unpolished surfaces**.
- GPU solver algorithm has been optimized to provide better **memory management** when running **multiple GPU simulations** in a row.
- The GPU solver algorithm has been improved to correctly take into account rays intersecting **Exit Geometries** (from Ray File or Surface source) during a simulation.
- The GPU solver algorithm has been aligned with the CPU solver algorithm for a better management of **Display sources** in simulation. Now **Fast Transmission Gathering** algorithm (when activated) automatically applies to the geometry (automatically created by the simulation) of a Display source. Thus, the geometry is ignored by the simulation.

Simulation result - HTML GPU Report

Now, GPU simulation report provides the same level of information as CPU simulation.

- **Propagation report** section has been added and includes the error report that provides the total number of errors and the list of error types.
- In addition, the **Performance Analysis** section of the report includes peak video memory usage that corresponds to the highest amount of vRAM consumed by the simulation at a certain time. This information becomes quite useful when you want to estimate how much the resolution of sensors can be increased for instance.



GPU acceleration - Rebalancing Camera simulation

Camera sensors in GPU Inverse simulations are now compatible with the Rebalancing mode set to **Variance** when running the Live Preview.

Virtual Photometric Lab / Virtual Human Vision Lab

- As of version 2026 R1, XMP files generated are in **version 10.3**. XMP files generated prior to version 2026 R1 will not load in the version 2026 R1 of Speos, Virtual Photometric Lab, and Virtual Human Vision Lab.
- XMP Filtering is now deactivated by default (set to None) for XMP results generated with a sensor without XML template applied. Consequently, simulations will require less memory during result generation and will deliver faster result output, enabling Virtual Photometric Lab / Virtual Human Vision Lab to open the results more quickly.



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

3. Changes

This section describes the operational and support changes introduced in Ansys SPEOS for Creo Parametric 2026 R1 release.

Ansys Product Improvement Program

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

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

Documentation

- Documentation has been updated with a new chapter explaining how to access and collect logs. This addition makes troubleshooting, debugging, and other diagnostic tasks easier. Besides, it also provides the development team with possible useful information to investigate and resolve software issues more efficiently.
- To improve clarity and provide a more coherent workflow, the Simulations and Results documentation chapters have been reorganized and refined.
  - **Restructured Simulation chapter**: The Simulation content is now organized into clearer, stage-based sub-sections that better reflect the progression of the simulation process.
  - **Results integrated into the workflow**: The former Results chapter has been moved under Simulation and renamed Analyzing Results, placing outcome information directly alongside the steps that produce it.
  - **Enhanced navigation**: The table of contents has been updated to reflect this new structure, making it easier for users to follow the end-to-end process.

Deprecated / End of support / End of life features

- **FTM Layer** is deprecated and in **end of support in version 2026 R1** and will be in **end of life in version 2027 R1**.
- **Solstis ray files** are deprecated and in **end of support in version 2026 R1** and will be in **end of life in version 2027 R1**.
- The following filtering options of Virtual Photometric Lab are deprecated and in **end of support in version 2026 R1**:



- The following filtering options of Virtual Human Vision Lab are deprecated and in **end of support in version 2026 R1**:

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



- The human vision Dynamic Adaptation option is deprecated and in end of support in version 2026 R1 in the following tools:

  – Virtual Human Vision Lab

  – Virtual Reality Lab

  – Virtual Lighting Animation Tool

  The default **Adaptation type** value is now set to **Dynamic Adaptation 2019**.

- The **Deterministic algorithm** with Photon Map (Build Photon Map, Load Photon Map, Build and Save Photon Map) is in **end of support in version 2026 R1**.
- The **Camera Post Processing** is in **end of support in version 2026 R1**.
- **Ansys Cloud Direct** is **deprecated on December 31, 2025**. Since this date, job submission to Ansys Cloud Direct are no longer supported. Also, Ansys Cloud Direct documentation is removed from the Ansys Help site in January.

Versions Update

- Windows: Windows 10 and Windows 11

  Minimum NVIDIA Driver **581.42** is required from 2026 R1.

- Speos GPU solver algorithm is now compatible with NVIDIA Blackwell GPU architecture.

Operating System

- Version of Creo 8 is no longer supported as of 2026 R1.
- Versions of Creo 9 from 9.0.1.0 to 9.0.11.0 are now supported as of 2026 R1.
- Versions of Creo 10 from 10.0.0.0 to 10.0.9.1 are now supported as of 2026 R1.
- Versions of Creo 11 from 11.0.0.0 to 11.0.6.0 are now supported as of 2026 R1.
- Versions of Creo 12 from 12.0.0.0 to 12.4.1.0 are now supported as of 2026 R1.

4. Resolved Issues and Limitations

For a list of issues that have been resolved in Release 2026 R1, refer to the Resolved Issues and Limitations document on the Ansys Help website.

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

Speos Labs Release Notes

The following sections present the new features and changes for Speos Labs 2026 R1 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 2026 R1 release.

Speos RPC

Refer to the Developer Documentation portal to see the latest changes for Speos RPC.

Simulation - new 3D Irradiance sensor result

Direct simulation with 3D irradiance sensor now generates a new result that contains all useful information for post processing. This is quite useful, especially when performing a Sun burn analysis.

Information for each 3D face includes:

- Every related to the meshing: facets, vertices and normals
- Radiometric values: irradiance, reflection, transmission, absorption, total power and absorbed power
- Data to advanced filtering: area and number of impacts per facet

This new result *.rawxm3 filenew result *.rawxm3 file uses the HDF5 format that can be opened with open-source software or via Python script.



Virtual 3D Photometric Lab

Upon creation of a new area in the Measures table, the purple area is automatically set to half the width and height sizes of the bounding box. In the Measures window, Dimension section, you can resize the Width and Height freely.



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

Virtual Photometric Lab / Virtual Human Vision Lab

- As of version 2026 R1, XMP files generated are in **version 10.3**. XMP files generated prior to version 2026 R1 will not load in the version 2026 R1 of Speos, Virtual Photometric Lab, and Virtual Human Vision Lab.
- XMP Filtering is now deactivated by default (set to None) for XMP results generated with a sensor without XML template applied. Consequently, simulations will require less memory during result generation and will deliver faster result output, enabling Virtual Photometric Lab / Virtual Human Vision Lab to open the results more quickly.



2. Changes

This section describes the operational and support changes introduced in Ansys Optical Labs 2026 R1 release.

Deprecated / End of support / End of life features

- **FTM Layer** is deprecated and in **end of support in version 2026 R1** and will be in **end of life in version 2027 R1**.
- **Solstis ray files** are deprecated and in **end of support in version 2026 R1** and will be in **end of life in version 2027 R1**.
- The following filtering options of Virtual Photometric Lab are deprecated and in **end of support in version 2026 R1**:



- The following filtering options of Virtual Human Vision Lab are deprecated and in **end of support in version 2026 R1**:



- The human vision Dynamic Adaptation option is deprecated and in end of support in version 2026 R1 in the following tools:
  - Virtual Human Vision Lab
  - Virtual Reality Lab
  - Virtual Lighting Animation Tool

  The default **Adaptation type** value is now set to **Dynamic Adaptation 2019**.

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

Versions Update

- Windows: Windows 10 and Windows 11

  Minimum NVIDIA Driver **581.42** is required from 2026 R1.

- Speos GPU solver algorithm is now compatible with NVIDIA Blackwell GPU architecture.

3. Resolved Issues and Limitations

For a list of issues that have been resolved in Release 2026 R1, refer to the Resolved Issues and Limitations document on the Ansys Help website.

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

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

Speos HPC Release Notes

The following sections present the changes and enhancements for Speos HPC 2026 R1 release.

1. New Features and Enhancements
2. Resolved Issues and Limitations

1. New Features and Enhancements

Simulation

- The multi-GPU memory-management algorithm has been optimized to deliver better performance when running simulations containing a Physical Camera sensor.
- The Speos HPC algorithm has been enhanced to provide faster simulation result-generation performance.

GPU

- Windows: Windows 10 and Windows 11

  Minimum NVIDIA Driver **581.42** is required from 2026 R1.

- Linux

  Minimum NVIDIA Driver **580.95.05** is required from 2026 R1.

- Speos GPU solver algorithm is now compatible with NVIDIA Blackwell GPU architecture.

2. Resolved Issues and Limitations

For a list of issues that have been resolved in Release 2026 R1, refer to the Resolved Issues and Limitations document on the Ansys Help website.

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

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

Ansys Acoustics Products

Release notes are available for the following Ansys Acoustics Products:

ANSYS Sound (p. 227)

Ansys Sound Release Notes

The following sections present the new features, enhancements and changes for Ansys Sound™ products in the 2026 R1 release.

1. Changes in Product Behavior
2. Ansys Sound: Analysis and Specification
3. Ansys Sound: Jury Listening Test

1. Changes in Product Behavior

Release 2026 R1 includes new features and enhancements that result in product behaviors that differ from previous releases. These behavior changes are presented below.

- The default **ECMA-418-2 Psychoacoustic Tonality** indicator in **Ansys Sound: Analysis and Specification** has been updated to use the 3rd edition of the standard instead of the 1st edition used in previous releases. Results for this indicator may therefore differ from those obtained using previous versions of the application.

2. Ansys Sound: Analysis and Specification

The following sections present the new features and enhancements for the Ansys Sound: Analysis and Specification™ application 2026 R1 release.

2.1. New Features

ECMA-418-2 Tonality Indicator

The ECMA-418-2 psychoacoustic tonality indicator has been updated to support the 3rd (2024) edition of the standard in addition to the 1st (2020), selectable using a drop-down menu in the UI. The 3rd edition is now the default setting, so results may differ from those obtained using previous versions of the application.

Sound Generation from dBA Spectrum

The AnsysSound_Spectrum file format has been incremented to version 4 to add support for dBA and dBA/Hz data.

Calculate Octave and One-third Octave Levels from PSD

Octave and One-third octave levels can now be calculated from a Power Spectral Density (PSD), in addition to a temporal signal.

Level vs Frequency File Format

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

A new file format has been created to allow import of files containing level against frequency for Generation of Harmonics from a Waterfall and Creating a Track with a Harmonics Source.

2.2. Enhancements

Mix Table

The Mix Table user interface has been updated to allow complete display of the signal names.

Spectrum Calculation Parameters

You can now update and reapply Spectrum Calculation Parameters for any signals displayed in a spectrum window.

2.3. Resolved Issues and Limitations

For a list of issues that have been resolved in Release 2026 R1, 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 2026 R1 release.

3.1. Enhancements

Recording Device Equalization

The setup for the Sounds Database has been restructured to allow selection of the appropriate recording device from an Equalization menu and a subsequent Setting for that device where applicable.

3.2. Resolved Issues and Limitations

For a list of issues that have been resolved in Release 2026 R1, refer to the Resolved Issues and Limitations document on the Ansys Help website.

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

Ansys Autonomous Vehicle Products

Release notes are available for the following Ansys Autonomous Vehicle products:

Ansys AVxcelerate Lighting and Sensors (p. 231)

Ansys AVxcelerate Sensors (p. 233)

AVxcelerate Lighting and Sensors Release Notes

The following section presents the enhancement and changes for AVxcelerate Lighting and Sensors 2026 R1 release.

1. Enhancement

2. Changes

1. Enhancement

This section describes the enhancement introduced in AVxcelerate Lighting and Sensors 2026 R1 release.

1.1. Co-simulation with CarMaker 14

You can now perform Co-simulation with CarMaker using version 14.1. The AVxcelerate Headlamp Library for CarMaker has been enriched with a sample project as well as TestRuns, roads and assets for this CarMaker 14 version.

AVxcelerate Lighting and Sensors is compatible with CarMaker versions 12.0.1, 13.0 and 14.1. CarMaker 11 is no longer supported.

1.2. Enhanced security added to gRPC communication

In AVxcelerate Lighting and Sensors, the default configuration now uses the UDS transport mode to secure the connection between the software and CarMaker.

For more details, refer to **Transport Modes** section on this page in the User Guide.

2. Changes

This section describes the changes introduced in AVxcelerate Lighting and Sensors 2026 R1 release.

2.1. Integrated Plugins

The Virtual Regulation plugins, previously delivered as add-on packages, are now included into the Ansys AVxcelerate Headlamp product. You can find the C-IASI, IIHS, and FMVSS components in the `VirtualRegulations` subfolder in the AVxcelerate Lighting and Sensors installation directory.

2.2. Updated Installation Paths

The following File Locations have been changed:

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

- The AVxcelerate Lighting and Sensors software is now installed in a the `Autonomy\AVxcelerateHeadlamp\VRX` folder, no longer in `Optical Products\VRXPERIENCE\VRX`.
- The AVxcelerate Lighting and Sensors logs are now saved to `%localappdata%\Ansys\v261\Autonomy\AVxcelerateHeadlamp\VRX`.
- The default location for SCANeR resources corresponding to your AVSimulation SCANeR™ version are now located in a folder named `AVX_SCANeR_2026R1` in the `AVSimulation` folder.

2.3. Windows 10 No Longer Supported

In the 2026 R1 release, all references to Windows 10 have been removed from the product documentation. This change aligns with Microsoft's official end of support for Windows 10, ensuring the product remains consistent with current platform standards and supported environments.

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

AVxcelerate Sensors Release Notes

The following sections present the new features, enhancements and changes for the AVxcelerate Sensors™ solution 2026 R1 release.

1. New Features
2. Enhancements
3. Changes

1. New Features

This section describes the new features introduced in AVxcelerate Sensors Simulator 2026 R1 release.

1.1. OSI Input Availability added to AVX Simulations

ASAM Open Simulation Interface (OSI) API is now available as an addition to the standard AVxcelerate Sensors Simulator API. This enhancement enables seamless integration with industry-standard simulation environments and toolchains, improving interoperability for ADAS and autonomous driving development workflows.

Benefits include:

- **Standardized Data Exchange:** Supports OSI-compliant environment data formats.
- **Improved Compatibility:** Easily connect with third-party simulation tools and ASAM-compliant platforms.
- **Future-Ready:** Aligns with automotive industry standards for simulation and testing.

For more details on ASAM OSI and how to configure it, refer to the Ansys Developer Portal.

1.2. IP Protection for Camera Sensors

Using the AVxcelerate Sensor Labs application or the Sensors REST API v8, you can activate IP protection on any section of a camera or fisheye camera, namely **Lens System**, **Imager**, **Default Post Processing** or **Sensor Output**. It is now also possible to activate the **Global Protection** in order to protect all the sections of the sensor at once.

The following parameters however are always visible and editable even when protection is activated:

- the **Windshield** parameters of the **Lens System** section,
- the **Native Resolution** parameter of the **Imager** section,

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

- the **Frame Rate**, **Time Encoding** and **Diffraction Spike** parameters of the **Default Post Processing** section,
- the **Output** parameter of the **Sensor Output** section.

2. Enhancements

This section describes the enhancements introduced in AVxcelerate Sensors Simulator 2026 R1 release.

2.1. Camera Simulation with Light Propagation Engine

First released in 2025 R2, the new rendering engine for camera sensor simulation remains a beta feature in 2026 R1. The performance as well as the output quality of camera simulations with Light Propagation Engine have been greatly improved thanks to new features and various enhancements.

2.1.1. New Raytraced Ambient Occlusion

Instead of using the Screen Space Ambient Occlusion to compute ambient occlusion maps, you can now use **Raytraced Ambient Occlusion**, which calculates shadows and depth more accurately, therefore producing more photo-realistic output.

Occlusion Map with Screen Space Ambient Occlusion



Occlusion Map with Raytraced Ambient Occlusion



2.1.2. New Raytraced Reflections

You now have the possibility to enable the new **Raytraced Reflections** feature so that the specular materials reflect not only the sky, but also surrounding objects taking into account the different types of light sources in the scene.

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



2.1.3. Improved Raytraced Shadows

When the **Raytraced Shadows** parameter is enabled, all the shadows are now raytraced, no matter the type of shadow configured in the properties of the point lights (static or dynamic). Thanks to this enhancement, GPU memory consumption remains moderate even when the scene includes many point lights. For more information on the above three ray-tracing features in LPE, see the Camera Simulation Parameters page in the User's Guide.

2.1.4. Enhancement to the Fog feature

The fog beta feature delivered in the previous release did not simulate atmospheric scattering effects. Now, fog color accounts for sky radiance, producing more realistic variations based on the camera's angle to the sun.

2.1.5. Rain in simulation

The 2026 R1 release introduces a real-time, physics-based rain phenomenon to enhance the realism of camera testing. The rain dynamically interacts with the environment and lighting conditions, improving accuracy in sensor evaluation.

In addition to the rain intensity, the wind conditions can be set. The rain rendering depends on the vehicle's speed and is dynamically correlated to the camera sensor exposure time.

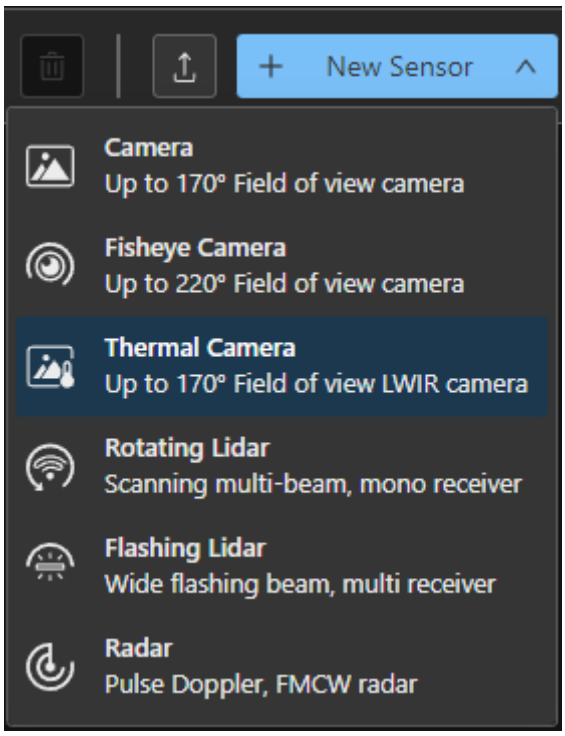
Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.



Note: Rain Simulation is released as a **beta feature**. Functionality may evolve in future versions.

2.2. Thermal Camera Sensor Creation and Edition

From now on you can create, edit and manage thermal cameras from the AVxcelerate Sensor Labs application or its REST APIs just like any other type of sensors.



New sets of dedicated parameters let you configure each component of a thermal camera sensor so that the simulated model reflects the specifications of your Long Wavelength InfraRed (LWIR) camera.

If safeguarding your Intellectual Property is a concern, you can activate the IP protection on one or several specific components, **Lens System**, **Imager**, **Default Post Processing**, or on the entire sensor. The only parameters that are always visible are the imager's native resolution and the frame rate.

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

After creation and edition, thermal camera models can be downloaded as a file (with the *lwircamerax* extension) or added to a sensor layout that will be used in simulation.

Important note: Be aware that thermal camera sensors from any version prior to 2026 R1 are now longer supported in the 2026 R1 release of the AVxcelerate Sensors.

The Thermal Camera Editor tool, which was previously delivered as a beta feature, as well as the packer and unpacker executables are no longer delivered.

2.3. GPU Direct Memory Access (DDMA) to Sensors Outputs

The GPU Direct Memory Access feature allows accessing raw sensor data directly on GPU, without going through the CPU & Memory. Fast GPU post-processing or fast data transfer for HIL injection are now possible.

Delivered as a beta feature, the DDMA Software Development Kit (SDK) includes the components required to develop your own custom library on Windows or Ubuntu operating systems, namely the DDMA extension, the *ProcessSensorData.h* header file, as well as samples (such as Camera GPU DDMA and Radar DDMA).

In addition, the new *GPU RDMA* sample, which is an alternative to the already existing *CPU RDMA* sample, allows for the transfer of some types of camera sensor output data from the GPU memory using an RDMA sender.

For more details, especially on the supported data formats, refer to Raw Data Output for DDMA and Raw Data Output for RDMA.

In this release, an additional step is required to use DDMA or RDMA, the **Data Access Settings** section in the simulation parameters of your radar or camera sensor must be configured with the new **Data Direct Memory Access** parameter, in which you can define the **Target** to either **GPU** or **CPU** according to the memory domain you want to use.

2.4. Co-simulation with CarMaker 14

AVxcelerate Sensors Simulator is now also compatible with CarMaker version 14.1.

The complete list of CarMaker and CarMaker HiL versions supported by AVxcelerate Sensors Simulator 2026 R1 are listed in AVxcelerate Sensors Simulator System Requirements.

2.5. Radar Simulation Improvement with Respect to Curved Surfaces

Part of the Radar Simulation Parameters, this optional parameter indicates whether or not to consider the curvature of a surface for bouncing rays. The curvature of the surface being always considered for the back scattered power calculation at the receiver antenna.

2.6. New Tracks Added

Two new tracks from 3D Mapping Solutions are available for the library, which correspond to real locations in Germany:

- **A8 Leonberg - Germany (highway):** This is included in the installation library.

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

- **Downtown Ingolstadt - Germany (urban):** Available for download at the Ansys Download Center, found on the *Current Release* page in the *AddOn Packages* section.

For other tracks corresponding to different locations, contact 3D Mapping Solutions at 3D Mapping. They will provide an environment in Ansys AVxcelerate .track data format with physics-based augmentation.

2.7. Addition of NCAP 2026 Assets and Track

NCAP (New Car Assessment Program) assets and track are now provided during product installation for use in simulations.

- Asset files provided are located in the [Library Installation Directory]/AVX_Library_v261/Assets_Standalone folder.
- Track file provided is located in the [Library Installation Directory]/AVX_Library_v261/Tracks_Standalone folder.

The dimensions of the NCAP assets are based on NCAP specifications. Refer to Euro NCAP AEB LSS VRU Test Protocol - v4.5.1.

2.8. Enhanced security added to gRPC communication used within the AVxcelerate Sensors™ solution

Enhanced security settings have been implemented to enable secure-by-default settings in gRPC.

2.8.1. AVxcelerate Sensors Simulator and API

To ensure security, AVxcelerate Sensors Simulator now uses the UDS transport for internal communication across its processes on the same workstation.

If you are distributing AVxcelerate Sensors Simulator processes across multiple workstations (see Multi-machine Simulations), AVxcelerate Sensors Simulator must be configured to use the mTLS transport mode instead of UDS.

To pilot AVxcelerate Sensors Simulator or to collect sensor data via its API, you have to select the appropriate transport mode depending on whether it is done from the same workstation or not:

- UDS transport mode when it is done from the same workstation,
- mTLS transport mode when it is done from a remote workstation.

An insecure option remains available, but its use is not recommended.

For more details, refer to Transport Modes.

2.8.2. Asset Preparation API

With the default communication configuration, the Asset Preparation server now only accepts connections from client applications using the Unix Domain Socket (UDS) transport mode.

You should then modify your client applications so that they use the same transport mode.

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

Alternatively, you may use the insecure transport mode.

For more details, refer to the Transport Modes section in the AVxcelerate Asset Preparation API Documentation.

3. Changes

This section describes the changes introduced in AVxcelerate Sensors 2026 R1 release.

3.1. AVxcelerate Sensors Simulator API v1

In the 2026 R1 release, the AVX internal gRPC server supports two versions of the AVxcelerate Sensors Simulator API: v0 and v1.

The AVxcelerate Sensors Simulator API v0 delivered in 2026 R1 is identical to the version delivered in 2025 R2. This version of the API will be officially discontinued in 2028.

In the AVxcelerate Sensors Simulator API v1, new features have been added and deprecated fields have been removed. The comprehensive list of changes can be found in the **Changelog** section of the API documentation available in the *AVxcelerateSensors/APIs/VSS_API/Documentation/v1* folder or on the Ansys Developer Portal.

To benefit from new features introduced in 2026 R1, any custom applications or scripts must be updated to use AVxcelerate Sensors Simulator API v1. Additionally, Ansys strongly recommends to update your simulation parameters json files.

The CarMaker connectors, the samples illustrating the usage of the AVxcelerate Sensors Simulator API, as well as the simulation parameters example files provided in the AVxcelerate Sensors Library or in the AVxcelerate Sensors Library for LPE now use the AVxcelerate Sensors Simulator API v1.

3.2. Discontinued Tools and Samples

- The packer and unpacker executables are no longer delivered.
- The *Binary Data Transfer* sample is discontinued.

3.3. AVSimulation SCANeR™ Connector

SCANeR connector and co-simulation is no longer distributed as part of the installer. It has to be requested from the support team.

3.4. Windows 10 No Longer Supported

In the 2026 R1 release, all references to Windows 10 have been removed from the product documentation. This change aligns with Microsoft's official end of support for Windows 10, ensuring the product remains consistent with current platform standards and supported environments.

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

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

Ansys Mesh Prep Products

Release notes are available for the following Ansys Mesh Prep products:

Meshing (p. 243)
ICEM CFD (p. 247)

Meshing Application Release Notes

Release 2026 R1 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. Topology Suppress
3. Morph Control
4. Enhancements to Mesh Workflows
5. Enhancements to MultiZone Method
6. Enhancements to Weld
7. Enhancements to Meshing Options
8. Enhancements to Sizing Group

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 **2026 R1** release includes the following changes in product behavior:

- In **Mesh Workflows**,
  - **Create Labels** operation is renamed to **Add Labels**.
  - **Proximity Label Creation** is renamed to **Proximity Label Addition**.
  - **Scope Label Creation** is renamed to **Scope Label Addition**.
- **Automatic (PrimeMesh)** method can now apply sizing before **Repair Topology** operations.

2. Topology Suppress

This is the first release of **Topology Suppress**. **Topology Suppress** allows you to suppress the selected entities in the topology. **Topology Suppress** supports only solids scoped with **Multizone** and shells scoped with **Automatic (PrimeMesh)**methods.

For more detailed information, see Topology Suppress.

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

3. Morph Control

This is the first release of **Morph Control**. **Morph Control** allows you to select points in the mesh, move the selected points with a prescribed transformation and perform a smooth interpolation for the morphable layer between the moving and the fixed nodes without destroying the mesh integrity.

For more detailed information, see **Morph Control**.

4. Enhancements to Mesh Workflows

Mesh Workflows has been enhanced to:

- Provide two new mesh workflows namely:
  - **FSI FEM Acoustics**
  - **Direct Morphing**
- **Stacker Mesh Workflow** has been enhanced to include:
  - **MultiZone: Mesh Surface** operation
  - **Create Solder Ball** operation
  - Non-stackable faces and edges outcome
  - Stacker diagnostics to check intersections in bodies.
  - Merge replicated bodies to the parent bodies in mesh volume.

  For more detailed information, see Stacker Mesh Workflow.
- Provide new controls and outcomes in operation.

For more detailed information, see Mesh Workflows.

5. Enhancements to MultiZone Method

MultiZone Method has been enhanced to perform smoothing on mesh using **Enable Smoothing** and skips smoothing on inflation layers when **Freeze Inflation Layers** is **Yes**.

For more detailed information, see MultiZone Mesh Method.

6. Enhancements to Weld

Weld has been enhanced to create **Weld Tail Extension** at both ends of the weld body when the **Source** is **Mesh**.

For more detailed information, see Weld.

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

7. Enhancements to Meshing Options

Meshing Options has been enhanced to include the following options:

- **Global Sizing Type** to choose the default behavior for **Mesh Sizing** under **Sizing Group** in **Mesh** Details view.
- **Show Hex Dominant Method for Explicit Physics** to display Hex Dominant Method for **Explicit** Physics Preference. The default value for **Show Hex Dominant Method for Explicit Physics** is **No** as **Explicit** Physics Preference does not prefer pyramid generation.
- **Mechanical Physics Preference Fallback on Failure** to enable fallback option when meshing fails on a part or body. The default value is **Adaptive Remesh**.
- **Mesh Disconnected Body Groups in Parallel** to perform parallel meshing on disconnected body groups. Here, body group is a collection of multiple bodies that are connected and is treated as a single entity for meshing. The default value is **No**.

For more detailed information, see Meshing Options.

8. Enhancements to Sizing Group

Sizing group has been enhanced for **Mechanical** Physics Preference to include **Mesh Sizing** to refine the mesh size based on the mesh sizing options. The available mesh sizing options are:

- **Coarse**
- **Medium**
- **Fine**
- **Uniform**
- **Advanced**
- **Adaptive**

For more detailed information, see Sizing Group.

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

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

ICEM CFD

Release 2026 R1 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 2026 R1 changes and new capabilities include the following:

1. Import Geometry Options

1. Import Geometry Options

Import Geometry now supports only the following **Legacy** CAD formats for both Windows and Linux OS:

- CATIA V4
- Rhino 3DM
- Plot3d

Import Geometry does not support the following **Legacy** formats:

- ACIS (Linux only)
- DWG (Linux only)
- GEMS, IDI (Linux only)
- ParaSolid (Linux only)
- STEP / IGES (Linux only)

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

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

Ansys Connect Products

Release notes are available for the following Ansys Connect products:

Ansys Minerva (p. 251)
Distributed Compute Gateway (p. 255)
optiSLang (p. 257)
ModelCenter (p. 265)

Ansys Minerva

The following topics provide an overview of what is new and improved in Ansys Minerva 2026 R1:

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 2026 R1:

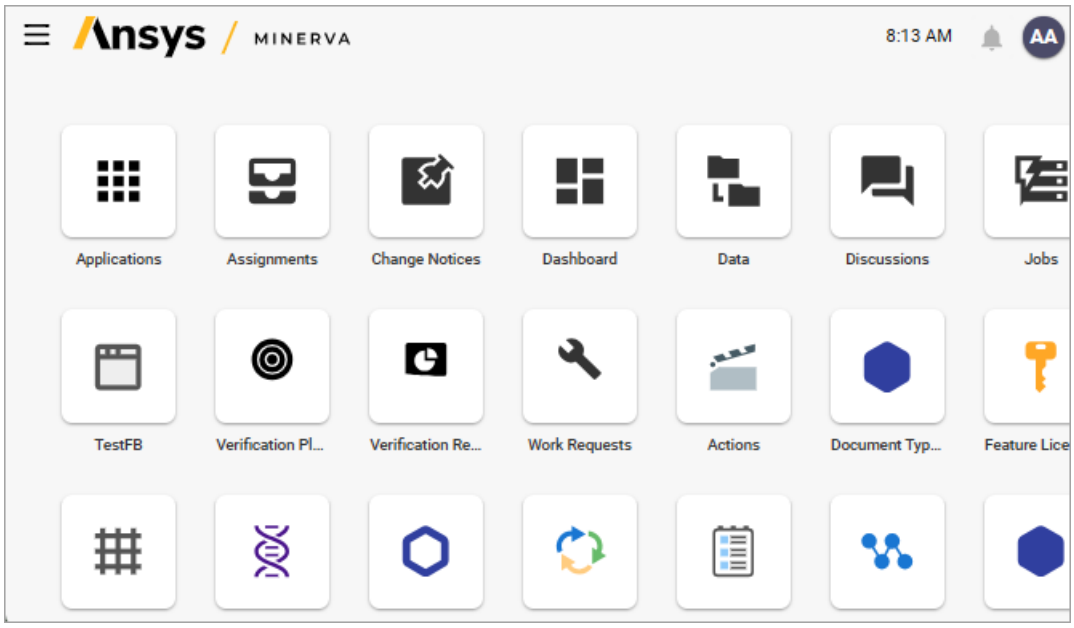
1.1. User Experience
1.2. Data
1.3. Configuration

1.1. User Experience

You now have an improved user experience when navigating Ansys Minerva, with the following changes:

- The overall user interface, including the Form 2.0 elements, has been updated to provide a more modern and consistent look.
- Less frequently used options have been hidden from the item toolbar and user menu.
- Unused applications such as **Paint**, **Notepad**, and **Remote Desktop** have been removed.

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



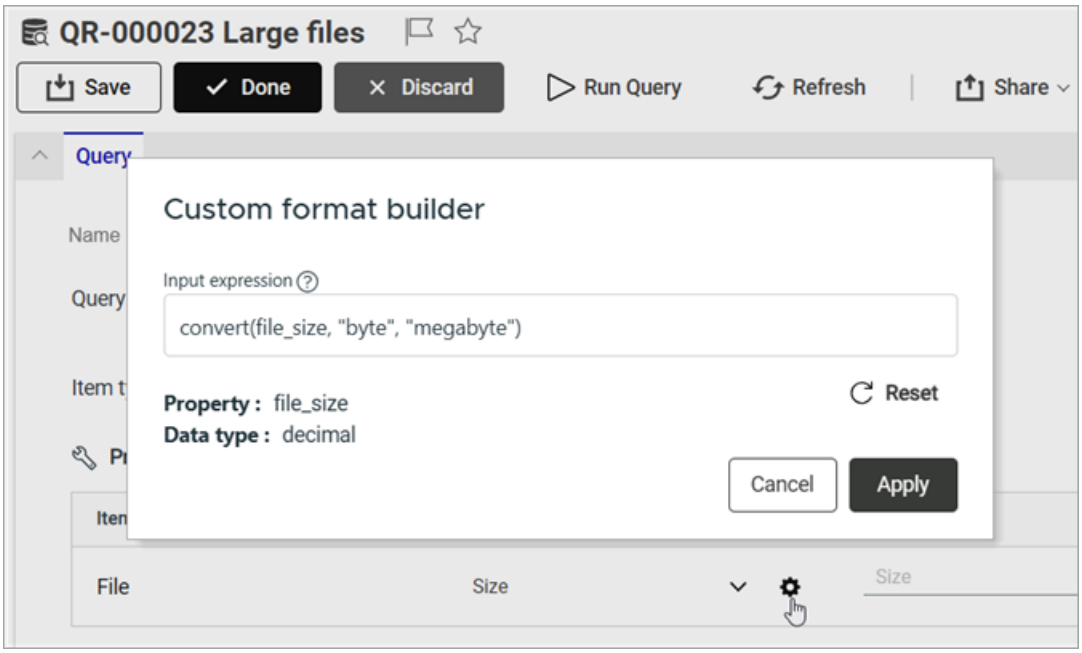
1.2. Data

- You can now open a folder in a new **Data** view tab for faster navigation and better content management. To do so, use either of the following methods:
  - In the **Data** view, select a folder and click **Open**, or **Ctrl**+ click the folder.
  - When viewing an item, click any of its parent folders in the breadcrumbs.
- When you select an item in the **Data** view and click **Properties**, the item now opens in a new tab with the **Properties** tab active. Previously, a fixed set of properties was displayed in a dialog.

1.3. Configuration

- In the query builder, when defining a custom format for the property information to be returned, you now need to use a Mamba expression instead of an SQL query. This helps you protect your data from risks like SQL injection.

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



For more information, see ????.

- The following enhancements have been made to Forms 2.0:
  - The **Name** property is now required for the **Form** element.
  - The styling of the form elements have been improved to provide a more modern look.
  - You can now configure an **Action Submit** button to validate the form on click. This prevents users from submitting the form with invalid inputs. See ????.
  - A new **Preview** button is available in the Form 2.0 editor. It enables you to preview the form with a desired context, exactly as it will appear when rendered.

2. Resolved Issues

For a list of issues that have been resolved in Ansys Minerva 2026 R1, 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 ???? 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*.

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

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*: ????.
- 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*: ????.

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

Distributed Compute Gateway (DCG)

The following topics provide an overview of what is new and improved in Distributed Compute Gateway 2026 R1:

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.

Advisory

Release 2026 R1 will be the final version of the Distributed Compute Gateway (DCG) application. Support for DCG will end in 2027.

The functionality of DCG 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.

For more information, see the **Ansys HPC Platform Services** page on the Ansys Help site or contact Ansys Support.

2. Resolved Issues

For a list of issues that have been resolved in Distributed Compute Gateway 2026 R1, 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*.

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

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

optiSLang

The following sections contain release information for Ansys optiSLang 2026 R1:

1. General
2. User Interface
3. Connectors
4. Algorithms

1. General

Installation Improvements: Installer Granularity & Size Reduction

- **Added New Install Options**
  - **Ansys optiSLang Core Headless** (1.5 GB) - For "inside" Flagships (except AEDT/Workbook) and for pyoptiSLang
  - **Ansys optiSLang Core UI** (0.9 GB) - Desktop UI, Integrations, Examples, tools
  - **AI** (2.2 GB)
  - **AI+ App Generation** (0.6 GB)
- Reduced total optiSLang installation size from 6.7GB to 1.5GB for "inside" use (78% reduction).
- Uses more Ansys central components

Ansys optiSLang "inside" Ansys Discovery

- Define variations now supports Sensitivity Analysis
  - Determines predictive quality and sensitivity matrix
  - Visualize the relationship between inputs and output with 3D response surfaces plots
  - Improves investigation of the influence of parameterized inputs on outputs
- One Click Optimization (coming in 2026R1 SP1)
  - Embedded optiSLang algorithms allow users to set constraints or objectives to optimize against
  - Ansys Discovery builds a Pareto front with provided range of parameters plus maximum quantity of variations
  - Easily explore a range of values before validating designs in Ansys Mechanical, Fluent, or Icepak

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

- Log/debug messages available in Ansys Discovery. Export to optiSLang GUI and Postprocessing for more.

PyOptiSLang Enhancements

Configuration Example

- Algorithms are directly accessible via pyoptiSLang. Define parametric and create design studies.
- Execute via Callbacks, run with Getter/Setter
- Inspect result designs
- Goto optiSLang

PyOptiSLang Cheat Sheet

- Added to the pyoptislang documentation.
- Similar to other PyAnsys Libraries
- Summarizes key functionality
- Covers: optiSLang instance management, project management, design evaluation, workflow generation, and working with placeholders.

SSH Remote Execution Enhancements

- Most changes require no "reset" of remote execution
- Remote profile changes are now reflected in remote execution
- Improved reconnect capabilities
- Now handles brackets in system name/working directory path
- C-Shell (csh/tcsh) remotes now fully supported
- Applying wizards to SSH systems now fully working
- Can now handle renamed systems/changed design directories
- Passwordless setup script now works for ports other than 22.
- Improved/extended logging

HPS Integration

- Self-contained optiSLang projects can be prepared and published to an HPS service instance.
  - The integration supports parameters, responses, placeholders, and input files defined at project level.

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

- HPS-published projects can then be used in Ansys optiSLang workflows
  - Change input parameters/files
  - Receive output parameters as responses/output files

2. User Interface

GUI Updates

- Supported UI styles on Windows:
  - Dark, Light
    → use fixed Ansys colors
    → Independent of any Windows color/style/theming settings
  - Fusion, Windows, Windows Vista, Windows11
    → Follows and respects the Windows platform palette.
    → Works with high-contrast themes in the Windows accessibility settings
- Fixed styling issues in Plugin Dialogs and HighDPI/4K issues in the overall GUI.

Workflow US Style Adoption

- Edges and flyouts now have rectangular corners, with a new setting for round corners on flyouts.
- In Condensed mode, flyouts no longer expand. The slot item itself is the drag-and-drop area.
- Postprocessing icons have been updated.

3. Connectors

Unified Run Options

- "Delay before execution" / "Delay between parallel executions" options are now consistent across integration nodes with new **Run Options** tab

LS-OPT Node

- New LS-OPT node allows you to use legacy LS-OPT projects in optiSLang,
- Workflows can be continued/extended with algorithms like OCO and AMOP.
- Integration & Automation can be accessed with more solvers or AI.
- Parent-Child licensing context for using the same license.

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

Ansys SimAI Solver Node (Beta)

- Inputs: Geometry, Boundary Conditions
- Outputs: Defined Global Coefficients, Confidence Score is shown
- Option to download result in vtk format

Ansys Rocky

- Updated the connector to single node solution
- Implemented guided solver wizard

Ansys FreeFlow

- New node based on Rocky integration
- Implemented guided solver wizard

Ansys Lumerical Node

- Added Parameter extraction and Output Extraction from all sources (Model, Root Element, Structure groups)
- Added Custom script path support (including relative paths to *.lsf)
- Added Automatic solver selection
- Now allows Script for Pre-/Post-Processing
- Added GPU-based execution
- Added Export tab for plots and data

Ansys OpticStudio Node

- Added a Radius vs. curvature setting
- Supports Hybrid mode (sequential mode with NSC and parameter extraction from both editors)
- Added Merit function value extraction
- Enhanced support for archive files
- Added an Export tab for plots and data

COMSOL Connector

- Extended support to COMSOL v3
- Adopted new customization interface v2520 standard

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

- Added the test run feature
- Added a tutorial for COMSOL

NX and NASTRAN Connectors

- Extended support to latest versions
- Adopted new customization interface v2520 standard
- Added a tutorial for NASTRAN

Fluent Mesh Node

- Added support for post-processing script to customize and adapt mesh postprocessing

ETK Abaqus

- Introduced ODB file version chooser: you can select the version when opening an ODB file
- Added a new column to the Variables tab to show the selected Abaqus version. The version can also be changed from here.
- Backward compatible to Abaqus 2021.

ETKnD Node

- Supports plugins: 2D Grid, CSV as point cloud, RST to VTP/VTU, RST to JSON, Images.
- Export format options: vtp, vtu, and sdb.
- Provides convenient and consistent data generation for 2D/3D AI training.

VTK Import in optiSLang 3D Postprocessor

- Available in oSP 3D GUI, oSL workflow, and API.
- Allows multi-design import with mesh mapping and single design/mesh import (via API).
- Supports .vtp and.vtu

Field Data Mining Node

- Prepare data for Machine Learning with support for optiSLang 3D PP and Twin Builder.
- Supports vtp, vtu, and sdb files.
- Added support for different mesh mappers and creating FMOP.

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

SimAI Integration

- Full workflow integration: Run DOE & process chain to create training data, upload and train for Ansys SimAI, use Ansys SimAI in optimization workflows.

4. Algorithms

Damped Least Squares (DLS) Algorithm (Beta)

- Added a new optimizer, Damped Least Squares (DLS) (also known as Levenberg-Marquardt), dedicated to calibration optimization tasks. Deterministic optimizer excellent for local optimization near optimum.
- Integrated in One-Click-Optimizer (OCO): Added Calibration mode in advanced options. Supports the following calibration expressions:

Expressions / Data	Signal	Vector	xy_data
euklidnorm(<data>)	Yes	Yes	Yes
norm(<data>,2)	Yes	Yes	Yes
<data>*<data>	n/a	Yes	n/a

- Provides stable one-click solution, independent of range and start point.

New Signal models

LSTM (Long Short-Term Memory)

- Parameters repeated and stacked with index "t" times.
- Predicts one element at a time until entire signal is predicted.
- Current output fed back into cell for memory behavior.

IndexNN

- Problem scalarized - every signal element modeled as single scalar problem.
- Index appended as parameter at input, NN predicts element at a time.
- The approximation function is responsible for reconstructing the initial signal

Benchmarks: Tested on 9 datasets with sensitivity analysis disabled, variance averaged CoP showing improved build time and quality.

Probability Function Export

- Available in Postprocessing. Statistical data plot visual.
- Syncs with histogram plot.

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

- Can export parameter definition (including correlation)
- Can import in Robustness/Reliability workflows.

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

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

ModelCenter Release Notes

The following sections contain release information for Ansys ModelCenter 2026 R1:

1. ModelCenter Desktop
2. ModelCenter Remote Execution
3. ModelCenter MBSE

1. ModelCenter Desktop

- Added support for Java 21. Java 8 support is now deprecated.
- Added support for the versions of CPython (3.10) and Java (Eclipse Temurin JDK 21) that ship with and are installed by the Ansys installer.
- Added support for SOLIDWORKS 2025.
- Added support for MATLAB 2025A.
- Added support for CATIA V5 2025 (R35).
- Added support for NX 2506.
- Added support for Creo 12.
- Added UI improvements, including a new icon set, and better support for high DPI scaling.
- Added Ansys Engineering Copilot.
- Added a Python script component plug-in (BETA feature).

2. ModelCenter Remote Execution

- Added support for Java 21. Java 8 support is now deprecated.
- Added support for the versions of CPython (3.10) and Java (Eclipse Temurin JDK 21) that ship with and are installed by the Ansys installer.
- Improved support for LDAP authentication.

3. ModelCenter MBSE

- Added support for Java 21. Java 8 support is now deprecated.

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

- Added support for the versions of CPython (3.10) and Java (Eclipse Temurin JDK 21) that ship with and are installed by the Ansys installer.
- Allow organizing execution plan results under specific plans in the Workspace Explorer.
- Added support for requirement attributes linked to constraints in the TeamWork Cloud Live Data plug-in.
- Added support for Capella 7.0.
- Added Ansys Engineering Copilot.
- Added support for expression evaluation in the MBSE connector for Ansys SAM (ALPHA feature).
- Added the ability to publish requirement validation status to Ansys SAM.
- Allow saving baseline values when attributes are inherited in the MBSE connector for Ansys SAM.

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

Ansys Platform Products

Release notes are available for the following Ansys Platform products:

ACT (p. 269)
Ansys Dynamic Reporting (p. 271)
CAD Integration (p. 275)
DesignXplorer (p. 279)
RSM (p. 281)
System Coupling (p. 283)
Workbench (p. 287)

ACT

No enhancements were made to ACT in 2026 R1.

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

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

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 2026 R1.

- First full release of the Ansys Dynamic Reporting Serverless configuration, allowing you to take advantage of ADR's capabilities without having to run an ADR-specific service.
- Added ability to export Python scripts for the Serverless configuration in the Template Editor.
- Added support for JSON template import/export in Serverless ADR.
- Updated the user interface libraries to modernize the look and feel of reports.
- Redesigned the three-way image comparison to improve the user experience.
- Added the Table Mapping generator template for mathematical manipulation of table values.
- Added an API to get the list of selected parts in the AVZ viewer component.
- Improved autoscaling for line plots and added font and font size control for PowerPoint templates.
- Added natural sorting to Data Filter templates.
- Added API to control the order of templates, with performance improvements in the Server configuration.
- Added properties to control the tick spacing in plots.
- Improved the user input validator.

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

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

Ansys Viewer

The following sections contain release information for Ansys Viewer 2026 R1:

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*.

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

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

CAD Release Notes

This section summarizes the new features in CAD Integration Release 2026 R1.

For more information, see the CAD Integration section of the Ansys Help.

Rocky Linux 8.10, 9.4, and 9.6 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 22.04 and 24.04 Support

The following interfaces are supported by Ansys:

- ACIS (Reader)
- AutoCAD (Reader)

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

- 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 2025 (Reader, Linux and Windows)
- CATIA V5 V5-6R2025 (Reader, Linux and Windows)
- CATIA V6 3DXML 4.3 (Reader, Linux and Windows)
- Creo Elements / Direct Modeling 20.8 (Plug-In, Windows only)
- Creo Parametric 12.4.0.0 (Reader, Linux and Windows)
- Inventor 2026 (Reader, Linux and Windows)
- NX 2506 (Reader, Linux and Windows)
- Solid Edge 2026 (Plug-In, Windows only)

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 (Customers > Support> More Support > Platform Support).

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

Distributed Compute Services (DCS)

No enhancements were made to DCS in 2026 R1.

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

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

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.

Replaced the graphics library used for Workbench charting. The content of existing charts remains the same, but how they are displayed has changed. Some chart properties have changed. 3D charts and pie charts are no longer supported.

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

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

Remote Solve Manager (RSM)

The following sections contain release information for Ansys Remote Solve Manager 2026 R1:

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 2026 R1, 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*.

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

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

System Coupling

The following features have been added to the Ansys System Coupling™ 2026 R1 release:

1. Participants
2. Mapping Accuracy
3. System Coupling in Workbench
4. Mapping Analysis Type
5. System Coupling Infrastructure
6. Data Model
7. Participant Library
8. Known Issues and Limitations

1. Participants

The following improvements have been made to System Coupling's participant coupling:

- Added the ability to use the Fluent GPU solver with System Coupling.
- If a FMU supports `CanGetAndSetFmuState` and `CanSerializeFmuState` (listed in the `Model-Description.xml` file), the FMU proxy participant now supports restarts.
- Ansys Electronics Desktop participants now keep licenses checked out for the duration of the coupled analysis by default. An option is available to disable this if desired.
- Added support for Fluent parameters `max-iterations-per-step` and `number-of-iterations` so that you can set the maximum number of iterations for steady state and transient Fluent cases.
- Added support for Fluent Flow Boundary coupling.
- Added LS-DYNA as a solver in the System Coupling ecosystem (**beta feature**).
- Upper and lower bounds can now be provided for Attributes.
- System Coupling Participant Library APIs for Fluent have been updated, and the older version deprecated. You can still use the old APIs by activating the hidden alpha feature using the command-line interface.

```
>>> DatamodelRoot().ActivateHidden.AlphaFeatures = True
CouplingParticipant[<fluent participant name>].UseNewAPIs = False
```

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

2. Mapping Accuracy

You can expect the following changes for profile-preserving mapping on surfaces when source and target data or both on face centroids:

- Faster and more robust mapping and weights calculation
- Preserve second-order accuracy
- Different mapped statistics on source side
- Changes to mapped values on target side
- Previous behavior can be restored by setting `AnalysisControl.AvoidDataReconstruction.SurfaceMapping` data model setting to *False*.

3. System Coupling in Workbench

Replaced the graphics library used for Workbench charting. The content of existing charts remains the same, but how they are displayed has changed. Some chart properties have changed. 3D charts and pie charts are no longer supported.

4. Mapping Analysis Type

Added a new Mapping analysis type, a special case of co-simulation. Used for file-type participants with static data.

5. System Coupling Infrastructure

The following improvements have been made to System Coupling's infrastructure:

- You can now open pre-created System Coupling examples that are ready to solve and explore.
- Improved the logic used when launching the Ansys Electronics Desktop executable.
- Now allow the same library/Instancing objects to be reused on multiple interfaces.
- Improved the compatibility with non-ANSI characters.

6. Data Model

A list of the changes made to System Coupling's data model is available in the Migrating Command-Line System Coupling Scripts section of the *System Coupling User's Guide*.

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

7. 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.

8. 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*.

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

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

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 2026 R1:

- New Ansys HPC Platform Services job definition settings have been added.
- Released the Maxwell to Motion coupling.
- Replaced the graphics library used for Workbench charting. The content of existing charts remains the same, but how they are displayed has changed. Some chart properties have changed. 3D charts and pie charts are no longer supported.
- You can now retain design point data when submitting design points to Ansys HPC Platform Services, then automatically or manually download that data to import back into the Workbench project.

2. Engineering Data Workspace

For Release 2026 R1, 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**

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

3. Injection Molding Data

For Release 2026 R1, there are no new feature updates for Injection Molding Data.

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

Embedded Software Products

Release Notes

Release notes are available for the following Ansys embedded software products:

Scade One (p. 291)
SCADE Suite (p. 297)
SCADE Display (p. 301)
SCADE Test (p. 305)
SCADE Architect (p. 311)
SCADE LifeCycle (p. 315)
SCADE Solutions for ARINC 661 Compliant Systems (p. 319)
SCADE Avionics Package (p. 323)

Scade One Release Notes

The following sections present the new features and enhancements for Scade One 2026 R1 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 2026 R1 release.

Improved Modeling and Testing Workflows

- **Multiple and detached diagrams**: organize complex models with better scalability for operators.
- **Model pragmas and annotations**: add information about model elements and requirements traceability.
- **UI/UX for test results**: get clearer insights and streamlined reporting with test results visualization in the **Job graph**, **Model explorer** and **test harnesses**.
- **Improved data type support in interactive debug**:
  - Chars
  - Groups
- **Simplified test harness with simulation data with sensor data type**: no need to connect the Data source to the set sensor block.
- **SCADE Test import**: migrate existing test cases into the Scade One environment.
- UI/UX enhancements:
  - **Discoverability**: interact and navigate in model using new shortcuts and actions.
  - **For test execution jobs**: multi test harnesses support.
  - When opening projects with dependencies.
- Performance improvements:
  - Opening and navigating in models.
  - Viewing and editing block diagrams.
  - Performance and reactivity of on-the-fly checks.

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

Configurable and Efficient Code Generation

- **Probes**: observe values during simulation and testing activities, and gain deeper visibility into generated code behavior, by monitoring internal flows with an observation structure.
- **Root interface as globals**: generate root operator I/Os and context as global C variables.
- Memory control in generated code (Beta): add constrains on the code generator to have more control on memory allocation.
- C standard:
  - SwanCG generates C99 code.
  - SwanCG is also compatible with later standards, including C11 and C23.
- Optimized generated code: general optimizations to achieve better performance.

Better Connectivity

- **Simulink import**: import Simulink® and Stateflow® models into the Scade One tool with customizable translation, including Stateflow® charts, MATLAB® variables, and Simulink subsystems.
- **Configurable SAM import**: adapt SAM data import to your needs:
  - Python library for custom import.
  - Personal access token support for authentication.
- **Improved FMI export and Python wrapper**: imported code (as project resources) and Linux support.
- Updates and enhancements for **PyScadeOne**:
  - Automatic installation of Python tools in a Python virtual environment.
  - Updates and refactorings in the model classes:
    - → Rename Model classes (OperatorDeclaration, OperatorDefinition)
    - → Predefined groups (**bar**, **concat**, **byname**, **bypos**)
    - → Predefined array operators (**flatten**, **pack**, **reverse**, **transpose**)
  - Support of:
    - → Memory control annotations
    - → Documentation and traceability pragmas
    - → Multiple diagrams
    - → Project resources and dependencies (read / create)
    - → Test module (test harness, **Data source**, **Oracle**) (read / create)

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

→ Model Check jobs

– State Machine navigation

Documentation

- **UI localization** and integrated help resources in Japanese.
- New language examples: DC Motor, Moving Average.

Installation

The **Scade One** and **SCADE Rapid Prototyper** software packages are downloaded and installed using the **Ansys Automated Installer** (Ansys ID required), providing a single installer for most Ansys tools, including SAM Enterprise desktop tools. The installer only downloads and installs necessary packages, supports Windows and Linux, and enables silent or download-only installation.

2. Limitations/Known Issues

This section describes the limitations and workaround solutions applying to Scade One 2026 R1 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)

Projects

Save will overwrite/update all projects, including all referenced projects (except libraries in **Scade One** installation directory that are considered read-only).

Modeler

The following language features/constructs are not supported by the modeler:

- Comments or pragmas 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 types editing is limited in the signal editor.

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

Debugger

- Variant types are read-only and not editable 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.
- If navigating in an operator instance under a **forward**, the current **forward** iteration is not visible in the navigation pane.

SCADE Importer

- The output model may be incorrect (due to incompatibilities or lack of equivalent language construct) when the input model contains:
  - Anonymous polymorphic structures (ex : {a: 'T}).
  - Imported numeric types.
  - Public declarations whose definition depends on private imported types, private enumerated types, or values of a private enumerated type.
  - Unsupported combinations of **activate** and **restart**.
  - **lxor**, **cast** or n-ary operators in size expressions.
  - Signals declared in a state and emitted in a weak transition of that state.
- The output model is correct, but the behavior is not the same as the SCADE model in the following cases (the importer raises a warning):
  - Synchro transitions are not correctly imported if there is a state that is both initial and final.
  - When partial iterators applied on **restart** or **activate .. every** operators.
- The output model triggers warnings in the following cases:
  - The model contains circular dependencies between modules.
  - There may be overlaps in the generated diagrams: using the **-diagram_ratio** option can reduce these overlaps.

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 Test Importer

Consult the SCADE Test Importer documentation to get information about limitations for each service.

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

SAM Importer

Consult the System Architecture Modeler Importer documentation to get information about limitations.

Documentation

UI localization and integrated help resources in Japanese: some parts can be given in English, including diagnostics and log messages.

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

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

SCADE Suite Release Notes

The following sections present the new features and enhancements for SCADE Suite 2026 R1 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 2026 R1 release.

AnsysGPT Assistant Integration

- New Ansys GPT assistant:
  - Ability to open AI-powered assistant in the GUI

SCADE Suite UX Improvements

- Easy access to Code generator tools without the need to change active project

SCADE Suite Design Verifier Improvements

- Ability to check Operators Input definition domain
- Ability to check Operator's contract (Guarantees)
- Ability to run Design Verifier executable in command line

SCADE Python API

- SCADE accepts and works with a Python virtual environment from GUI and batch

SCADE Automotive Package

- **New SCADE ACG 2.8**:
  - Support of AUTOSAR parameters mapped to sensors
  - New certification credits for ISO 26262 (see Certification Credits (p. 298))
- Ability to set code generation using either ACG 2.6 or ACG 2.8, both with certification credits

SCADE Test Integration

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

For details, consult SCADE Test Release Notes (p. 305).

SCADE LifeCycle Integration

For details, consult SCADE LifeCycle Release Notes (p. 315).

2. Certification Credits

The SCADE Suite 2026 R1 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** and **SCADE ACG 2.8** for certification under ISO 26262 up to ASIL D
- See also Certification Credits (p. 306) for SCADE Test and Certification Credits (p. 315) for SCADE LifeCycle

3. Limitations and Workarounds

This section describes the limitations and workaround solutions applying to the SCADE Suite 2026 R1 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.

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

Limitation in Multicore Code Generator

- With clocked tasks, generated code may contain cyclic dependency between methods, which prevents from integrating it.

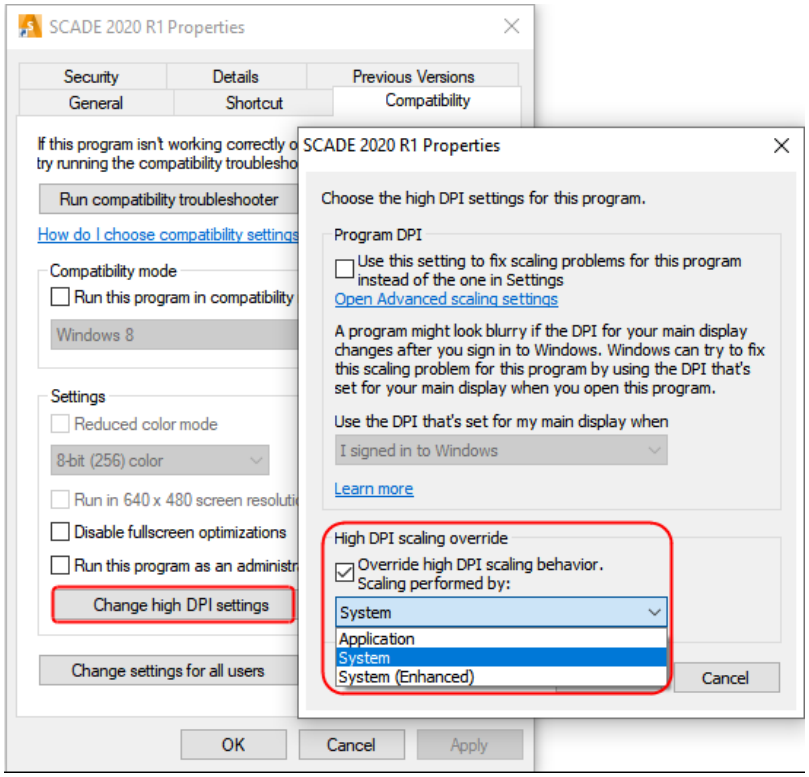
Limitation in SCADE Applications

- 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.

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

4. Migration Information

SCADE Suite 2025 R2 to 2026 R1

- No format change: no project migration issue

SCADE Automotive Package 2025 R2 to 2026 R1

- No format change: no project migration issue: see SCADE Automotive Package (each release) (p. 300)

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*.

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

SCADE Display Release Notes

The following sections present the new features and enhancements for SCADE Display 2026 R1 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 Model Export

- Ability to export model as FMU for Ansys STK integration
- Support a transparent overlay of the Display panel in STK views, as well as a dedicated window

SCADE Display UX Improvements

- Multi-spec batch snapshot export
- New command to display file directory containing project resources

SCADE Test Integration

For details, consult SCADE Test Release Notes (p. 305).

SCADE LifeCycle Integration

For details, consult SCADE LifeCycle Release Notes (p. 315).

2. Certification Credits

The SCADE Display 2026 R1 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

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

- See also Certification Credits (p. 306) for SCADE Test and Certification Credits (p. 315) for SCADE LifeCycle

3. Limitations and Workarounds

This section describes the limitations and workaround solutions applying to the SCADE Display 2026 R1 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.

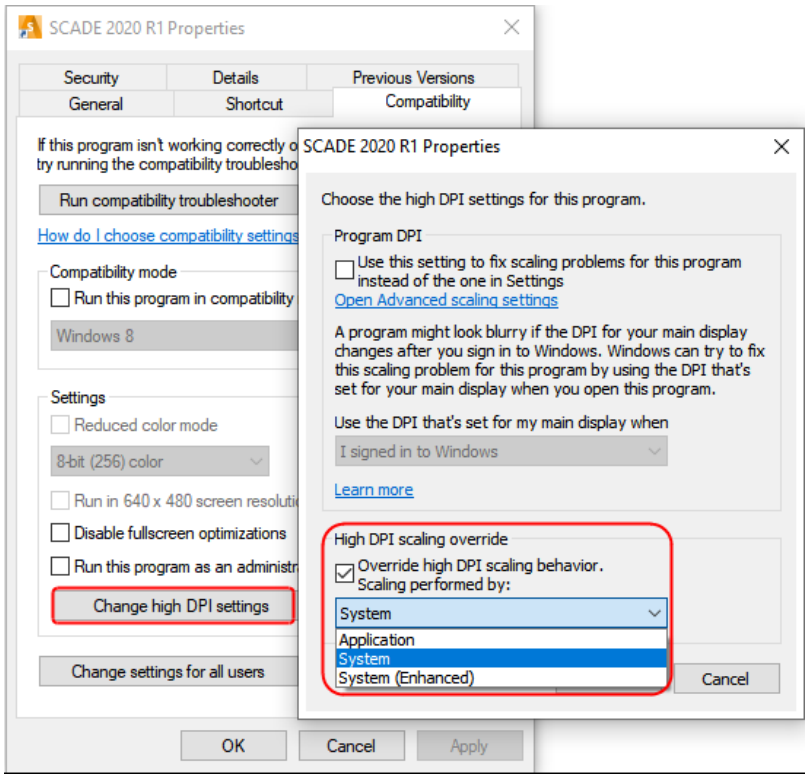
Limitation in SCADE Applications

- 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.

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



* 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 R2 to 2026 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*.

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

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

SCADE Test Release Notes

The following sections present the new features and enhancements for SCADE Test 2026 R1 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 2026 R1 release.

SCADE Test UX improvements

- Ability to check test cases without launching them

SCADE Python API

- SCADE accepts and works with a Python virtual environment from GUI and batch

Test Verifier

- New Test Verifier tool:
  - Ability to automate Test Case reviews when Test Cases are automatically generated for LLR Testing (for DO-178C DAL A Applications)

SCADE Test Target Execution for SCADE Display

- New VectorCAST support for SCADE Test for Display
- New certification credits (see Certification Credits (p. 306))

SCADE Test Environment for SCADE Display

- New certification credits (see Certification Credits (p. 306))

SCADE Test Environment for SCADE Display

- New command to display file directory containing project resources
- Ability to draw in frame buffer instead of window, with transparency
- Ability to export model as FMU for Ansys STK integration

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

SCADE LifeCycle Integration

For details, consult SCADE LifeCycle Release Notes (p. 315).

2. Certification Credits

The SCADE Test 2026 R1 release includes the following tool with certification credits:

- **SCADE Test Environment for Host for SCADE Display - 2026 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 Target Execution for SCADE Display - 2026 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 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 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)

3. Limitations and Workarounds

This section describes the limitations and workaround solutions applying to the SCADE Test 2026 R1 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

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

- 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)

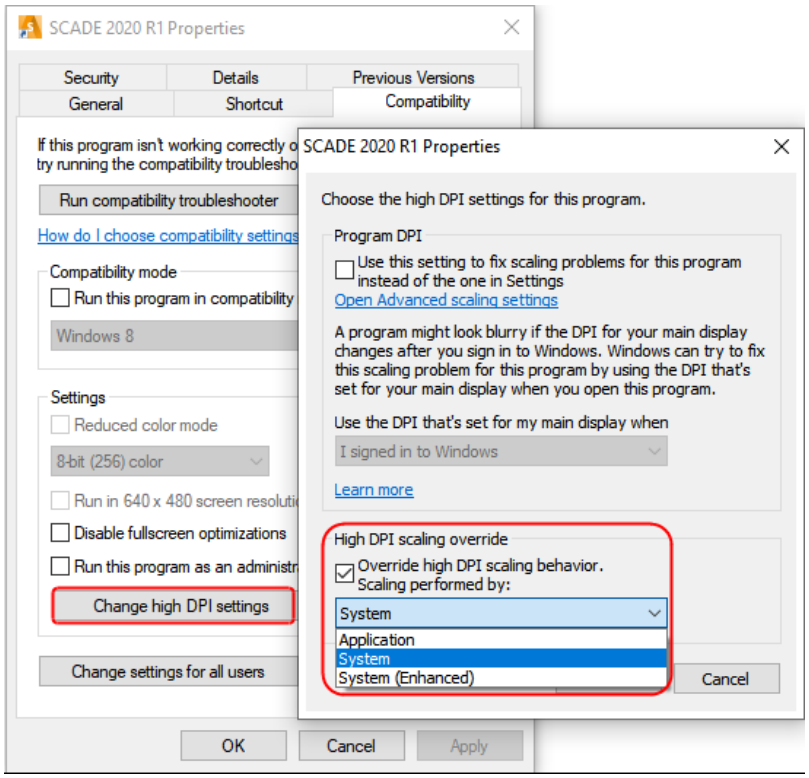
Limitation in SCADE Applications

- 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.

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



* 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 R2 to 2026 R1

- No format change: no project migration issue

SCADE Rapid Prototyper 2025 R2 to 2026 R1

- No format change: no project migration issue

For migration information related to previous versions, contact the Technical Support at https://customer.ansys.com.

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

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*.

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

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

SCADE Architect Release Notes

The following sections present the new features and enhancements for SCADE Architect 2026 R1 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. 315).

2. Limitations and Workarounds

This section describes the limitations and workaround solutions applying to the SCADE Architect 2026 R1 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.

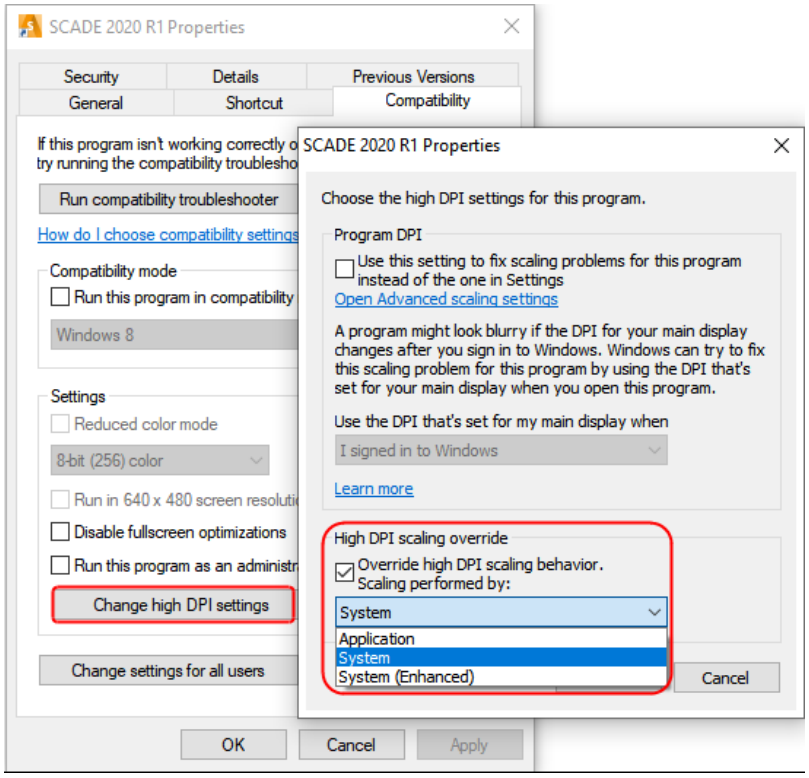
Release 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

- 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.

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

- 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 in-compatible 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 R2 to 2026 R1

- No specific issue: see SCADE Architect (each release) (p. 313)

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.

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

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*.

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

SCADE LifeCycle Release Notes

The following sections present the new features and enhancements for SCADE LifeCycle 2026 R1 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 2026 R1 release.

SCADE LifeCycle ALM Gateway

- Ability to import attributes in DNG Connector

2. Certification Credits

The SCADE LifeCycle 2026 R1 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**

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

  - 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 2026 R1 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.

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

- **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 R2 to 2026 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*.

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

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

SCADE Solutions for ARINC 661 Compliant Systems Release Notes

The following sections present the new features and enhancements for SCADE 2026 R1 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 2026 R1 Solutions for ARINC 661 release.

ARINC 661 Widgets

- Various maintenance updates improving stability and consistency

SCADE UA DF Generator

- **New SCADE UA DF Generator 6.8.1**:
  - New certification credits for DO-178C/DO-330 (see Certification Credits (p. 320))
  - Support hard-coded A661_BROADCAST_DATA message in BufferFormat widget
  - New option **-no_timestamp** to disable printing date in generated files
  - Support fr type with max ULONG for 3D widgets introduced in A661-8

Test Automation Framework (TAF)

- Updated documentation with a modern layout, now available exclusively in HTML
- Clarification of documentation on default image comparison
- Scope expansion to a complete, Python-based test environment for CDS and UAs

Error and Logging UX Improvements

- Improve control on server debug traces (via the command line)
- Finalize descriptiveness improvement of error messages

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

- Parse & report OGLX errors in UA Page Creator

SCADE LifeCycle Integration

- For details, consult SCADE LifeCycle Release Notes (p. 315).

2. Certification Credits

The SCADE 2026 R1 Solutions for ARINC 661 release includes the following tool with certification credits:

- **SCADE UA DF Generator 6.8.1**: for qualification under DO-178C/DO-330 up to TQL-1
- See also Certification Credits (p. 315) 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 2026 R1 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, EditBoxOpenEventExtension, 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

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

- 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

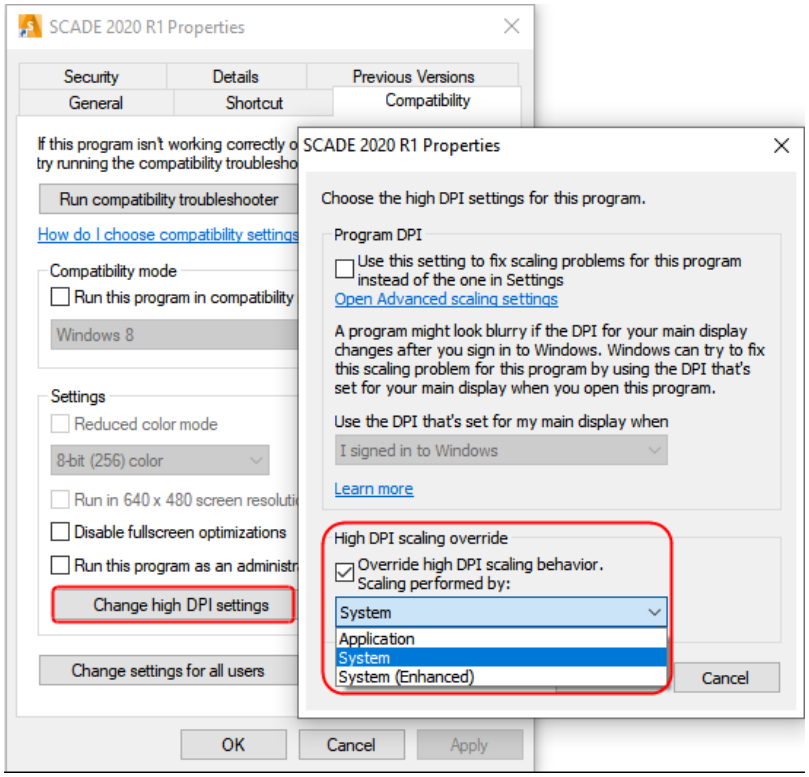
Limitation in SCADE Applications

- 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

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

- 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 R2 to 2026 R1 Solutions for ARINC 661

- No format change: no project migration issue

For migration information related to previous versions, contact the Technical Support at https://customer.ansys.com.

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

SCADE Avionics Package Release Notes

The following sections present the new features and enhancements for SCADE Avionics Package 2026 R1 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

- Management of extra operator I/Os corresponding to no FACE connections

2. Limitations and Workarounds

This section describes the limitations and workaround solutions applying to the SCADE Avionics Package 2026 R1 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.

- WString in FACE 2.1 are not supported by synchronization with SCADE Suite.

  Workaround: use BoundedString instead.

  .

  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.

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

Workaround: set Adobe Acrobat as default PDF reader.

3. Migration Information

SCADE Avionics Package 2025 R2 to 2026 R1

- No specific issue: see SCADE Avionics Package (each release) (p. 324)

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.

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

For migration information related to previous versions, contact the Technical Support at https://customer.ansys.com.

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

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