



POWERING INNOVATION THAT DRIVES HUMAN ADVANCEMENT

© 2024 ANSYS, Inc. or its affiliated companies
Unauthorized use, distribution, or duplication is prohibited.

Release Notes



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

Release 2024 R2
July 2024

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

Copyright and Trademark Information

© 2024 ANSYS, Inc. Unauthorized use, distribution or duplication is prohibited.

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

Disclaimer Notice

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

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

U.S. Government Rights

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

Third-Party Software

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

Published in the U.S.A.

Table of Contents

Global	xv
1. Advisories	xv
2. Compatibility with Previous Releases	xvi
3. Installation	xvi
4. Licensing	xvi
5. Documentation	xvii
6. Verification Manuals	xvii
6.1. Mechanical APDL Verification Manual	xviii
6.1.1. New Test Cases	xviii
6.1.2. Modified Test Cases	xviii
6.1.3. Removed Test Cases	xviii
6.2. Workbench Verification Manual	xviii
6.2.1. New Verification Test Cases	xviii
6.2.2. Modified Test Cases	xviii
6.2.3. Removed Verification Test Cases	xviii
6.3. Ansys LS-DYNA Verification Manual	xviii
6.3.1. LS-DYNA SOLVE Test Cases	xviii
6.3.2. LS-DYNA EMAG Test Cases	xix
6.3.3. Modified LS-DYNA Test Cases	xx
6.3.4. Removed LS-DYNA Test Cases	xx
7. Online Video Access	xx
8. Ansys Customer Site	xx
Ansys 3D Design Products	1
Discovery Release Notes	3
1. Discovery Product Advisories	3
2. Discovery Enhancements Overview	4
2.1. Fluid Flow	4
2.1.1. Fluid-Fluid Interfaces	4
2.1.2. Fluid Flow Meshing Method Default	5
2.2. Structural	6
2.2.1. Structural Solver (Explore)	6
2.2.2. Connection Assessment Tool (Explore)	6
2.2.3. Contact and Interface Review	6
2.3. Meshing	7
2.3.1. Local Fidelity in Explore	7
2.3.2. Mesh Display for Explore Fluids with Local Fidelity	7
2.3.3. Mesh Quality Metrics	8
2.4. Results Monitors	10
2.4.1. Output Parameters	10
2.4.2. Mass Flow-Weighted Average for Monitors	10
2.4.3. Surface Heat Flux Monitor	11
2.4.4. Reaction Resultant Monitors	12
2.4.5. Bolt Monitors (Explore)	13
2.5. Results Enhancements	13
2.5.1. Additional Fluid Emitters	13
2.5.2. Improvements to Results Display	14
2.5.3. Enhanced Messages for Results Confidence	14
2.5.4. Exporting Saved Scenes Snapshots	15
2.5.5. Results on Multiple Cut Planes	15

2.6. Hybrid Modeling	15
2.7. Shrinkwrap	16
2.8. History Tracking	16
2.9. Modeling Tools	17
2.9.1. Improvements to Modeling Documentation in Discovery	17
2.9.2. Hide and Show Faces	17
2.9.3. Clip Volume	17
2.9.4. Clip Plane	18
2.9.5. Equation Tool	18
2.9.6. Shared Topology Review	19
2.10. Materials	19
2.10.1. Availability of Materials Tools	19
2.10.2. Materials Data for Simulation	19
2.10.3. Materials Data for Simulation - Sample Library	21
2.11. Tree and Advanced Selection Panel	21
2.12. File Settings	22
2.12.1. GLB Export	22
2.12.2. File Options	22
2.13. User Interface Enhancements	24
2.14. Extension Builder Enhancements	24
2.14.1. Copy Script Reference and Close All Script Tabs Button	24
2.14.2. Text Box Enhancements	25
2.15. User Settings Summary	26
2.15.1. Mouse Settings	26
2.15.2. Check for Updates	26
2.15.3. Top View Direction	26
2.15.4. Custom Shortcuts	27
2.15.5. Improved Color Maps	27
2.16. Performance	28
2.16.1. Responsiveness Improvements	28
2.16.2. Background Automation for Physics	28
2.16.3. Display Support	28
2.17. Installation and Licensing	28
2.17.1. Installation Memory Requirement	28
2.17.2. HPC Requirement for LiveGX in CPU Mode	29
2.17.3. Command Line Option for -geometryservice Packet	29
2.17.4. Installation Prerequisite Checking	29
2.17.5. Web Named User Licensing	29
2.18. Online Help Enhancements	30
2.19. Ansys Discovery Technology Showcase: Example Problems	30
2.20. Ansys 3D Design Verification Manual	30
3. Discovery Caveats and Known Issues	31
SpaceClaim Release Notes	33
1. SpaceClaim Product Advisories	33
2. SpaceClaim Enhancements Overview	33
2.1. Scripting	33
2.2. Import Support	33
2.3. Meshing	34
3. SpaceClaim Caveats and Known Issues	34
DesignModeler Release Notes	35
1. DesignModeler Product Advisories	35

2. DesignModeler Enhancements	35
3. DesignModeler Caveats and Known Issues	35
Ansys Structural Products	37
Mechanical Application Release Notes	39
1. Changes in Product Behavior	39
2. Documentation	42
3. Performance	43
4. Add-ons	43
5. General	44
6. Graphics	45
7. Geometry	45
8. External Model	45
9. Contact and Connections	46
10. Mesh	46
11. Additive Manufacturing Process Simulation	46
12. Acoustics Analyses	47
13. Coupled Field Analyses	47
14. Analysis	47
15. Fracture	48
16. Structural Optimization	48
17. Motion	49
18. Loads/Supports/Conditions	49
19. Imported Loading Conditions	50
20. Solution	51
21. Explicit Dynamics	51
22. Results	52
Mechanical APDL	55
1. Structural	55
1.1. Contact	55
1.1.1. Enhancements to Contact Friction Models	56
1.1.2. Enhancements to Multipoint Constraints and Assemblies	56
1.1.3. DOF-based Rigid Surface Constraints and Rigid Elements	56
1.1.4. Performance Improvements to CNCHECK Command	56
1.1.5. Performance Improvements for Fluid Penetration Pressure Loads	57
1.1.6. Performance Enhancements related to Contact Surface Wear	57
1.2. Elements and Nonlinear Technology	57
1.2.1. New Genb Joint Element	57
1.2.2. Mesh-Coarsening Enhancements for Nonlinear Adaptivity	57
1.2.3. Element Removal for Nonlinear Adaptivity	58
1.2.4. Direct Element Embedding Enhancement	58
1.2.5. Automatic Mesh Optimization of Base Mesh	58
1.2.6. Preprocessing Stress Evaluation of Shell Elements	58
1.2.7. CPT Element Support for Reinforcing	58
1.2.8. Improved Output Control for Selected Result Data	58
1.2.9. Gasket Element Enhancements	58
1.3. Material and Fracture Modeling	59
1.3.1. Python Extension for Ansys Materials Lab	59
1.3.2. Material Curve-Fitting Enhancements	59
1.3.3. Initial Deformation for Hyperelastic Materials	60
1.3.4. Cyclic-Loading Enhancements	60
1.3.5. Inline Material Criterion for Adaptive Crack Initiation	60

1.3.6. von Mises Plasticity Enhancement	60
1.3.7. Support for Rotated Nodal Coordinate Systems in SMART Remeshing Zone	60
1.3.8. Support for Node and Element Components Tracking and Updating During SMART Remeshing	61
1.4. Linear Dynamics	61
1.4.1. Harmonic Balance Method (HBM) Cyclic Analysis	61
1.4.2. Fixed-interface CMS Method for Single Stage Sector Models	61
2. Multiphysics	61
2.1. Acoustics	61
2.1.1. Harmonic Nonlinear Acoustics Solver Model	62
2.1.2. Difference Frequency Method	62
2.2. Thermal	62
2.2.1. 3D Thermal Layered Shell Element: SHELL294	62
2.3. Coupled-Field	62
2.3.1. Enhanced Strain Formulation Supported for PLANE222 and SOLID225 for Structural- Electric-Diffusion Analysis	63
2.3.2. Output Flux Density Vectors for Coupled-Diffusion Analyses	63
2.3.3. Circuit Modeling Enhancement	63
3. Solvers	63
3.1. Distributed-Memory Parallel Processing Enhancements	63
3.2. Enhanced Memory Reduction for Models with Constraint Equations and High Core Counts	64
3.3. Reduced Peak Memory Usage for Uncompressed Node and Element Numbering	64
3.4. Support for Devices for the GPU Accelerator	64
3.5. Improved Performance for the PCG Solver	64
3.6. Support for Mixed u-P Formulation with the PCG Solver	64
3.7. Beta Feature - Frequency Domain Decomposition for Modal Analyses	65
4. Other Enhancements	65
4.1. Enhancement for Element-Single Value Results	65
4.2. CAD Support Updates	65
5. Commands	65
5.1. New Commands	65
5.2. Modified Commands	66
5.3. Undocumented Commands	67
6. Elements	67
6.1. New Elements	68
6.2. Modified Elements	68
6.3. Undocumented Elements	69
7. Documentation	69
7.1. Updates for Programmers	69
7.2. Feature Archive	69
8. Technology Showcase: Example Problems	69
9. Mechanical APDL Release 2024 R2 Update Guide	70
9.1. Backward Compatibility	70
9.2. Feature Updates Causing Result or Behavior Changes	70
9.2.1. Change in Default PCG Preconditioner for Static and Full Transient Single-Field Struc- tural Analyses	71
9.2.2. Change in automatic constraint type detection logic for bonded contact	71
9.2.3. Change in behavior when issuing LSCLEAR,ALL	71
9.3. Known Incompatibilities	71
9.4. Known Issues	71

9.5. Resolved Issues and Defects	71
9.6. Features Removed or Archived	72
Autodyn	73
1. New Features and Enhancements	73
Aqwa	75
1. Aqwa Solver Modules	75
2. Aqwa Graphical Supervisor (AGS)	75
3. Hydrodynamic Analysis Systems	75
Ansys Composite PrepPost (ACP) Release Notes	77
1. New Features and Enhancements	77
1.1. Support for Unicode	77
1.2. Composite Failure Tool in Mechanical	77
1.3. PyDPF Composites	78
1.4. Composite Failure Criteria Migration from ACP (Post) to Mechanical	78
1.5. Sectional Properties	78
1.6. Scripting of Composite Failure Criteria in Mechanical	78
1.7. CAD Geometries	78
1.8. Enhanced Definition of Reference Directions	78
1.9. Lay-up Extrusion of Solid Models	79
1.10. Bounding Box	79
1.11. PyACP (Beta) Joining PyAnsys	79
1.12. VTK Upgrade	79
1.13. Retirement of ACP (Post)	79
2. Known Limitations	79
Material Designer Release Notes	81
1. New Features and Enhancements	81
1.1. Short Fiber Composite Models	81
2. Known Limitations	81
Additive Manufacturing	83
1. DED Process Add-on	83
2. Sintering Process Add-on	83
3. Changes in Product Behavior - Additive in Mechanical	83
Sherlock	85
1. Known Issues	85
2. New Features and General Enhancements	85
3. API Updates	86
4. Bug Fixes	87
5. Part Library	90
Forming	91
Motion Release Notes	93
1. Changes in Product Behavior	93
2. New Features	93
3. Functional Enhancements	94
Ansys Fluids Products	97
Fluent	99
1. Supported Platforms for Ansys Fluent 2024 R2	99
2. New Features in Ansys Fluent 2024 R2	99
2.1. Meshing Mode	99
2.2. Solution Mode	100
2.3. Fluent Workspace Applications	109
3. Updates Affecting Ansys Fluent 2024 R2 Code Behavior	111

3.1. Meshing Mode	112
3.2. Solution Mode	112
3.3. Fluent Workspace Applications	114
4. Features To Be Removed in a Future Release	114
CFX	117
1. Supported Platforms	117
2. New Features and Enhancements	117
CFD-Post	119
1. Supported Platforms	119
2. New Features and Enhancements	119
3. Incompatibilities	119
4. Updates Affecting Code Behavior	120
TurboSystem Release Notes	121
1. Supported Platforms	121
2. New Features and Enhancements	121
TurboGrid	123
1. Supported Platforms	123
2. New Features and Enhancements	123
BladeModeler	125
1. Supported Platforms	125
2. BladeGen	125
2.1. New Features and Enhancements	125
3. BladeEditor	125
3.1. New Features and Enhancements	125
4. BladeBuilder	125
4.1. New Features and Enhancements	125
Polyflow	127
1. New Features	127
2. Supported Platforms	127
3. Resolved Issues and Limitations in Ansys Polyflow 2024 R2	127
Forte	129
1. New Features and Enhancements	129
2. Resolved Issues and Limitations	130
Chemkin	131
1. New Features and Enhancements	131
2. Resolved Issues and Limitations	132
FENSAP-ICE	133
1. New Features and Enhancements in Ansys FENSAP-ICE	133
2. Resolved Issues and Limitations	134
EnSight	135
1. New Features and Enhancements in Ansys EnSight	135
2. PyEnSight	136
3. Ansys Dynamic Reporting (Nexus)	137
4. Beta in This Release	137
5. Alpha in This Release	138
6. Resolved Issues and Limitations	138
Ansys Electronics Products	139
Icepak	141
1. Introduction	141
2. New and Modified Features in Ansys Icepak 2024 R2	141
3. Resolved Issues and Limitations in Ansys Icepak 2024 R2	141

Ansys Optical Products	143
Speos Release Notes	145
1. New Features	145
2. Enhancements	148
3. Changes	150
4. Resolved Issues and Limitations	151
Speos for NX Release Notes	153
1. New Features and Enhancements	153
2. Changes	156
3. Resolved Issues and Limitations	157
Speos for Creo Parametric Release Notes	159
1. New Features and Enhancements	159
2. Changes	161
3. Resolved Issues and Limitations	162
Speos Labs Release Notes	163
1. New Features and Enhancements	163
2. Changes	163
3. Resolved Issues and Limitations	163
Speos HPC Release Notes	165
1. New Features and Enhancements	165
2. Resolved Issues and Limitations	165
Ansys Acoustics Products	167
Ansys Sound Release Notes	169
1. Ansys Sound: Analysis and Specification	169
1.1. New Feature	169
1.2. Enhancements	169
1.3. Resolved Issues and Limitations	169
2. Ansys Sound: ASDforEV	169
2.1. New Feature	170
2.2. Enhancements	170
2.3. Resolved Issues and Limitations	170
3. Ansys Sound: Car Sound Simulator	170
3.1. Resolved Issues and Limitations	170
4. Ansys Sound: Jury Listening Test	170
4.1. New Features	171
4.2. Resolved Issues and Limitations	171
5. Ansys Sound: VR Sound	171
5.1. New Features	171
5.2. Resolved Issues and Limitations	171
Ansys Autonomous Vehicle Products	173
AVxcelerate Lighting and Sensors Release Notes	175
1. Enhancements	175
1.1. Co-simulation with AVSimulation SCANer™ 2023.2	175
1.2. Co-simulation with CarMaker 12.0.1	175
2. Changes	175
2.1. Shared Web Licensing	175
2.2. GPU Driver	176
2.3. Deprecated Sensors Removed	176
AVxcelerate Sensors Release Notes	177
1. New Features	177
1.1. IP Protection for Radar Models	177

1.2. Adaptive Grid Sampling in Radar Simulation	177
2. Enhancements	178
2.1. Co-simulation with AVSimulation SCANer™ 2021.1	178
2.2. Co-simulation with CarMaker 12	178
2.3. Improvements to LiDAR Model	178
2.4. States in AVxcelerate Asset Preparation API	179
3. Changes	180
3.1. Shared Web Licensing	180
3.2. Sensor Output Management	180
3.3. AVxcelerate Sensor Labs User Interface Changes	180
3.4. Power Emitted by Surface Sources	181
3.5. GPU Driver	181
3.6. VSS API Changelog	181
3.6.1. Changes to the Simulation service	181
3.6.1.1. configuration.proto	181
3.6.1.2. simulation_parameters.proto	182
3.6.2. New LightingSystem service	182
3.6.3. Changes to the Resource Uploader service	182
3.6.3.1. upload_data.proto	182
3.7. Asset Preparation API Changelog	183
3.7.1. AVxcelerate Asset Preparation API Server	183
3.7.2. AVxcelerate Asset Preparation API v3	183
3.7.2.1. Changes to the Geometry service	183
3.7.2.1.1. geometry.proto	183
3.7.2.1.2. geometry_description.proto	183
3.7.2.2. Changes to the Scene Tree service	184
3.7.2.2.1. scenetree_description.proto	184
3.7.2.3. Changes to the Environment service	186
3.7.2.3.1. environment_description.proto	186
3.7.2.4. Sky service removed - Natural Sky Preparation and HDRI Sky Preparation services added	186
3.7.2.5. Changes to the Material Preparation service	187
3.7.2.5.1. material.proto	187
3.7.2.5.2. material_description.proto	187
3.7.2.6. Changes to the Surface Source service	188
3.7.2.6.1. surface_source.proto	188
3.7.2.6.2. surface_source_description.proto	188
3.7.2.7. Changes to the Directional Light service	189
3.7.2.7.1. directional_light.proto	189
3.7.2.7.2. directional_light_description.proto	189
3.7.2.8. Changes to the Point Light service	190
3.7.2.8.1. point_light.proto	190
3.7.2.8.2. point_light_description.proto	190
3.7.2.9. Changes in the tag.proto file	191
Ansys Mesh Prep Products	193
Meshing	195
1. Changes in Product Behavior from Previous Releases	195
2. Geometry Fidelity	196
3. Enhancements to Automatic (PrimeMesh)	196
4. Enhancements to Mesh Workflows	196
5. Enhancements to Tetrahedrons Method	196

6. Enhancements to Feature Detection	197
7. Enhancements to Mesh Quality Worksheet	197
8. Enhancements to Weld Control	197
9. Enhancements to MultiZone Method	197
10. Enhancements to Quad Layer	197
11. Enhancements to Mesh Copy	198
12. Enhancements to Meshing Options	198
13. Enhancements to Mapped Meshing	198
14. Enhancements to Repair Topology	198
15. Enhancements to Sweep Method	198
ICEM CFD	199
1. Enhancements to CGNS Version	199
Ansys Connect Products	201
Ansys Minerva	203
1. New Features and Enhancements	203
1.1. Data	203
1.2. Verification Plans	205
1.3. Work Requests and Tasks	206
1.4. Generic Connector	206
2. Resolved Issues	207
3. Known Issues and Limitations	207
4. Documentation	207
Distributed Compute Gateway (DCG)	209
1. New Features and Enhancements	209
2. Resolved Issues	209
3. Known Issues and Limitations	209
optiSLang	211
1. Ansys Web App	211
2. Student Version of optiSLang	211
3. General	211
4. User Interface	212
5. Connectors	212
6. Algorithms	212
Ansys Platform Products	215
ACT	217
Ansys Viewer	219
1. New Features and Enhancements	219
2. Resolved Issues and Limitations	219
3. Known Issues and Limitations	219
CAD Release Notes	221
Distributed Compute Services (DCS)	223
DesignXplorer	225
Remote Solve Manager (RSM)	227
1. New Features and Enhancements	227
2. Issues Resolved in this Release	227
3. Known Issues and Limitations	227
System Coupling	229
1. Point Cloud for Co-Simulation	229
2. Electric Arc Modeling	230
3. Functional Mock-Up Unit (FMU) Co-Simulation Participants	230
4. Postprocessing	230

5. Graphical User Interface	230
6. Data Model and Commands	230
7. Known Issues and Limitations	231
Workbench	233
1. General Workbench Enhancements	233
2. Engineering Data Workspace	233
3. Injection Molding Data	234
Embedded Software Products	235
SCADE Suite Release Notes	237
1. New Features and Enhancements	237
2. Certification Credits	238
3. Limitations and Workarounds	238
4. Migration Information	240
5. Supported External Tool Versions	241
SCADE Display Release Notes	243
1. Fixes and Enhancements	243
2. Certification Credits	243
3. Limitations and Workarounds	244
4. Migration Information	245
5. Supported Graphical Libraries Version	245
SCADE Test Release Notes	247
1. New Features and Enhancements	247
2. Certification Credits	248
3. Limitations and Workarounds	248
4. Migration Information	250
5. Version Compatibility for Testing Environment	251
SCADE Architect Release Notes	253
1. Fixes and Enhancements	253
2. Limitations and Workarounds	253
3. Migration Information	255
4. Supported External Tool Versions	256
SCADE LifeCycle Release Notes	257
1. New Features and Enhancements	257
2. Certification Credits	257
3. Limitations and Workarounds	258
4. Migration Information	259
5. Supported External Tool Versions	260
SCADE Solutions for ARINC 661 Compliant Systems Release Notes	261
1. New Features and Enhancements	261
2. Certification Credits	262
3. Limitations and Workarounds	262
4. Migration Information	263
SCADE Avionics Package Release Notes	265
1. Fixes and Enhancements	265
2. Limitations and Workarounds	265
3. Migration Information	265

List of Figures

1. Learning and Support Ribbon Tab 106

2. Example: Folder Versioning ?

3. Example: Folder Versioning 205

Global Release Notes

The release notes are specific to Ansys, Inc. Release 2024 R2 and arranged by application or product, with the exception of:

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

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 2024 R2 can be found in the online version of [2024 R1 release notes](#). Or via [ansys.com](#)> Customer Center> Customer Center> General Support> Download Center> Downloads> Product Documentation> Select Release> PDF packages> release.

1. Advisories

- Ansys Distributed Compute Services (DCS) 2024 R1 was the final version of DCS released by Ansys. Support for DCS will end in July of 2025.

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

You should now be using HPC Platform Services instead of DCS. Existing DCS users can continue to use DCS if necessary but are strongly encouraged to transition to HPC Platform Services at their earliest convenience. The look and feel of HPC Platform Services will be very familiar to DCS users, making the transition quick and easy.

To get started or learn more, see the [Ansys HPC Platform Services](#) page on the Ansys Help site.

- 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 **Known Issues and Limitations** document. See the [Ansys customer site \(p. xx\)](#) 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 2024 R2, refer to the [Resolved Issues and Limitations](#) document on the Ansys Help site.
- Ansys Help (<https://ansyshelp.ansys.com/>) is not supported on the Internet Explorer browser.

2. Compatibility with Previous Releases

Backwards Compatibility: Ansys 2024 R2 was tested to read and resume databases from the following previous versions: 2022 R2, 2023 R1, 2023 R2, and 2024 R1. Note that some products are able to read and resume databases from releases prior to 2022 R2. See the specific product sections below for more information. For those products that cannot directly read an earlier database in 2024 R2, first resume it in a supported version and then resume that database in 2024 R2.

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

3. Installation

The following features are new or changed at Release 2024 R2. Review these items carefully.

- The Ansys Installation program includes support of RedHat 9 and Ubuntu 22.
- The Ansys Installation program includes support for Speos for NX 2406.
- The Ansys Installation program no longer includes support for Speos for Creo Parametric 6.
- The Ansys Installation program no longer includes support for Speos for NX 2206.
- The Ansys Installation program no longer supports the .NET CASPOL Utility.

4. Licensing

The following enhancements were made to Ansys, Inc. Licensing for Release 2024 R2:

- Web Licensing now includes new Named User products for Discovery Modeling and Discovery Simulation. For more information, see [Discovery Web-based Licensing](#) in the Discovery Installation Guide.
- Web Licensing now includes more products for Shared Web, including "Ansys HPC Pack", "Ansys AVxcelerate", "Ansys Scade One", "Speos for Creo Parametric", "Ansys medini", and several others. Consult your Ansys account representative for the complete list.
- Added support for users belonging to multiple user accounts in Web Licensing.
- The ID Portal now supports bulk import of lists of users via Excel and CSV files.
- The Web Assignments and Transactions table column widths are automatically resized based on contents.
- Included a new Web Assignments user interface for both Shared and Named User.
- The overall licensing experience has been enhanced to provide improved performance, stability and security.
- The Licensing Portal Export Active Server history has been updated to include all servers.

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 2024 R2:

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

6.1. Mechanical APDL Verification Manual

6.1.1. New Test Cases

No test cases were added for 2024 R2.

6.1.2. Modified Test Cases

- [VM97](#) was modified to include four-node thermal shell element [SHELL294](#).
- [VM271](#) was modified to include four-node thermal shell element [SHELL294](#).

6.1.3. Removed Test Cases

No test cases were removed for 2024 R2.

6.2. Workbench Verification Manual

6.2.1. New Verification Test Cases

No test cases were added for 2024 R2.

6.2.2. Modified Test Cases

No test cases were modified for 2024 R2.

6.2.3. Removed Verification Test Cases

No test cases were removed for 2024 R2.

6.3. Ansys LS-DYNA Verification Manual

Test Case Sets:

- [6.3.1. LS-DYNA SOLVE Test Cases](#)
- [6.3.2. LS-DYNA EMAG Test Cases](#)
- [6.3.3. Modified LS-DYNA Test Cases](#)
- [6.3.4. Removed LS-DYNA Test Cases](#)

6.3.1. LS-DYNA SOLVE Test Cases

- [VM-LSDYNA-SOLVE-022](#): **Statically Indeterminate Reaction Force Analysis**: An assembly of three prismatic bars is supported at both end faces and is axially loaded with forces F1 and F2. Find the reaction forces in the Y direction at the fixed supports.
- [VM-LSDYNA-SOLVE-023](#): **I-Shape Beam Deflection Under Uniformly Distributed Loads**: A standard WF beam is loaded on the overhangs by a uniformly distributed load. Determine the deflection at the middle of the beam.

- **VM-LSDYNA-SOLVE-024: Deflection of a Hinged Support:** A structure consisting of two equal steel bars with hinged ends is subjected to the action of a load. Determine the deflection of the hinged end.
- **VM-LSDYNA-SOLVE-025: Laterally Loaded Tapered Support Structure:** A cantilever beam tapers uniformly, with the large end attached to a wall. It is loaded by a force at the tip. Find the maximum bending stress at the fixed end of the beam.
- **VM-LSDYNA-SOLVE-026: Radial Displacement of a Pinched Cylinder:** A thin-walled cylinder is pinched by a force at the middle of the cylinder length. Determine the radial displacement at the point where force is applied.
- **VM-LSDYNA-SOLVE-027: 3D Taylor Cylinder Impact:** The Taylor cylinder impact test uses a right circular cylinder made of a test material which impacts a theoretically rigid target. The deformed cylinder after impact will be compared with the modeled results from the reference.
- **VM-LSDYNA-SOLVE-028: 2D Taylor Cylinder Impact:** The Taylor cylinder impact test uses a right circular cylinder of a test material which impacts a theoretically rigid target. In this test, an OFHC copper cylinder impacts a rigid plate. The deformed cylinder after impact will be compared with the modeled results from reference.
- **VM-LSDYNA-SOLVE-029: Heat Transfer from a Cooling Spine:** A steel cooling spine extends from a wall that is maintained at a given temperature. The tip of the spine is insulated. Find the heat conducted by the spine and the temperature of the tip.
- **VM-LSDYNA-SOLVE-030: Bending of Long Plate Subjected to Moment - Plane Strain Model:** A long, rectangular plate is fixed along the longitudinal face, and the opposite face is subjected to a moment of 5000 lbf-in about the Z axis. Find the X normal stress at a distance of 0.5 inches from the fixed support as well as the total deformation and reaction moment.
- **VM-LSDYNA-SOLVE-031: Deflection of Beam using Symmetry and Anti-Symmetry:** A long bar with simply supported ends is subjected to lateral load of 1000 N at a distance of 8 m from one end. Find the deformation in the Y direction at 8 m from the simply supported end. A symmetrical and asymmetrical test case is considered.
- **VM-LSDYNA-SOLVE-032: Fundamental Frequency of a Simply-Supported Beam:** Determine the fundamental frequency of a simply supported beam of 80 in.
- **VM-LSDYNA-SOLVE-033: Combined Bending and Torsion of Beam:** A vertical bar is subjected to the action of a horizontal force acting at a distance from the axis of the bar. Determine the maximum principal stress.

6.3.2. LS-DYNA EMAG Test Cases

- **VM-LS-DYNA-EMAG-009: Hollow Conducting Sphere in a Uniform Magnetic Field:** A hollow conducting sphere is in a uniform field that varies sinusoidally with time. Determine the real and imaginary values of the magnetic field along a given axis.
- **VM-LS-DYNA-EMAG-010: Asymmetrical Conductor with a Hole:** A thick aluminum plate with a hole, which is placed eccentrically and set unsymmetrically in a non-uniform magnetic field. Determine the real and imaginary parts of the magnetic flux between the coil and plate and compare with reference results.

- **VM-LS-DYNA-EMAG-011: Long Cylinder in a Uniform Sinusoidal Field:** An infinitely long cylinder has a uniform field of 0.1 Tesla at 60 Hz that is applied perpendicular to its axis. The objective of the test is to measure the amplitude and phase of the magnetic field along the Y axis.

6.3.3. Modified LS-DYNA Test Cases

No test cases have been modified for 2024 R2.

6.3.4. Removed LS-DYNA Test Cases

No test cases have been removed for 2024 R2.

7. Online Video Access

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

8. Ansys Customer Site

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

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

Ansys customer site access points:

- **Tutorials and input files**

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

- **Documentation**

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

- **General information**

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

Ansys 3D Design Products

Release Notes

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

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

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

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

Discovery Release Notes

Release 2024 R2 of Discovery contains the following.

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

1. Discovery Product Advisories

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

Installation and Licensing Advisories

- The documented memory requirement for installing Discovery has been changed to 16 GB RAM or greater. Previously, it was documented as 32 GB RAM or greater.
- 2024 R2 will be the last release to support the **Legacy Named User Subscription** option. It will be replaced with web-based Named User licensing at 2025 R1.
- In general, Discovery maintains support of cards for as long as NVIDIA drivers and SDKs allow. For a list of NVIDIA cards no longer supported at 2024 R2, see [Installation Prerequisites](#).

SpaceClaim Advisories

- SpaceClaim 2024 R2 will be the last version based on ACIS. You should plan your migration to Discovery.
- SpaceClaim will be removed from the Ansys unified installers at Release 2025 R1.

File Import and Export Support Advisories

For export, support for CATIA V5 Assembly and CATIA V5 Part have been dropped.

Discovery Beta Features

For information about Beta features available in Discovery 2024 R2, see *Discovery Beta Features Documentation* in the Ansys Help.

Also see [Discovery Enhancements Overview \(p. 4\)](#) for features that have been added or enhanced in Release 2024 R2 of Discovery.

2. Discovery Enhancements Overview

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

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

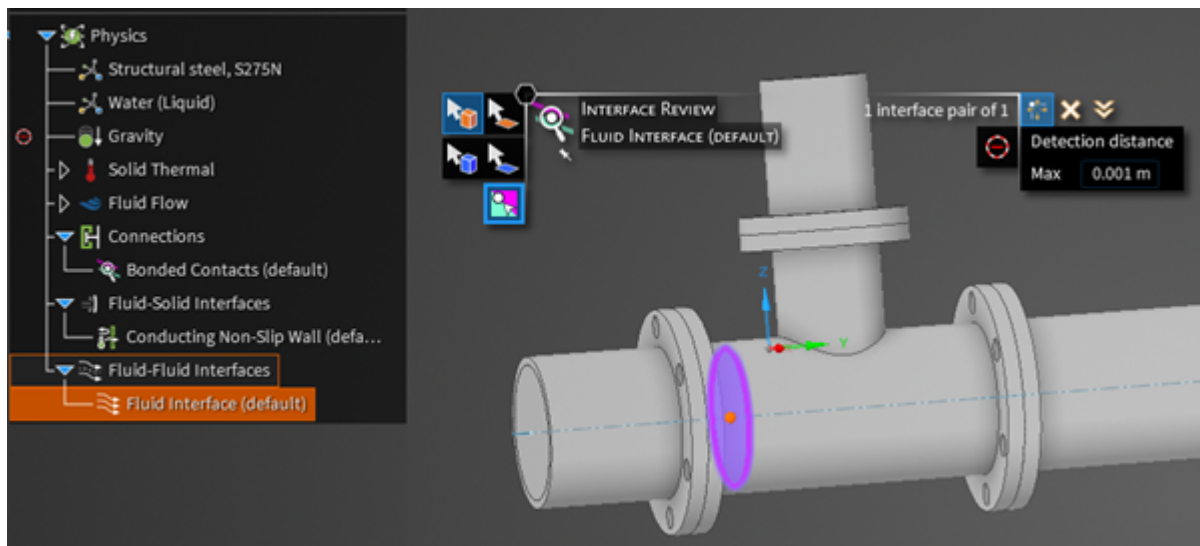
- [2.1. Fluid Flow](#)
- [2.2. Structural](#)
- [2.3. Meshing](#)
- [2.4. Results Monitors](#)
- [2.5. Results Enhancements](#)
- [2.6. Hybrid Modeling](#)
- [2.7. Shrinkwrap](#)
- [2.8. History Tracking](#)
- [2.9. Modeling Tools](#)
- [2.10. Materials](#)
- [2.11. Tree and Advanced Selection Panel](#)
- [2.12. File Settings](#)
- [2.13. User Interface Enhancements](#)
- [2.14. Extension Builder Enhancements](#)
- [2.15. User Settings Summary](#)
- [2.16. Performance](#)
- [2.17. Installation and Licensing](#)
- [2.18. Online Help Enhancements](#)
- [2.19. Ansys Discovery Technology Showcase: Example Problems](#)
- [2.20. Ansys 3D Design Verification Manual](#)

2.1. Fluid Flow

The following enhancements related to fluid flow are available in 2024 R2.

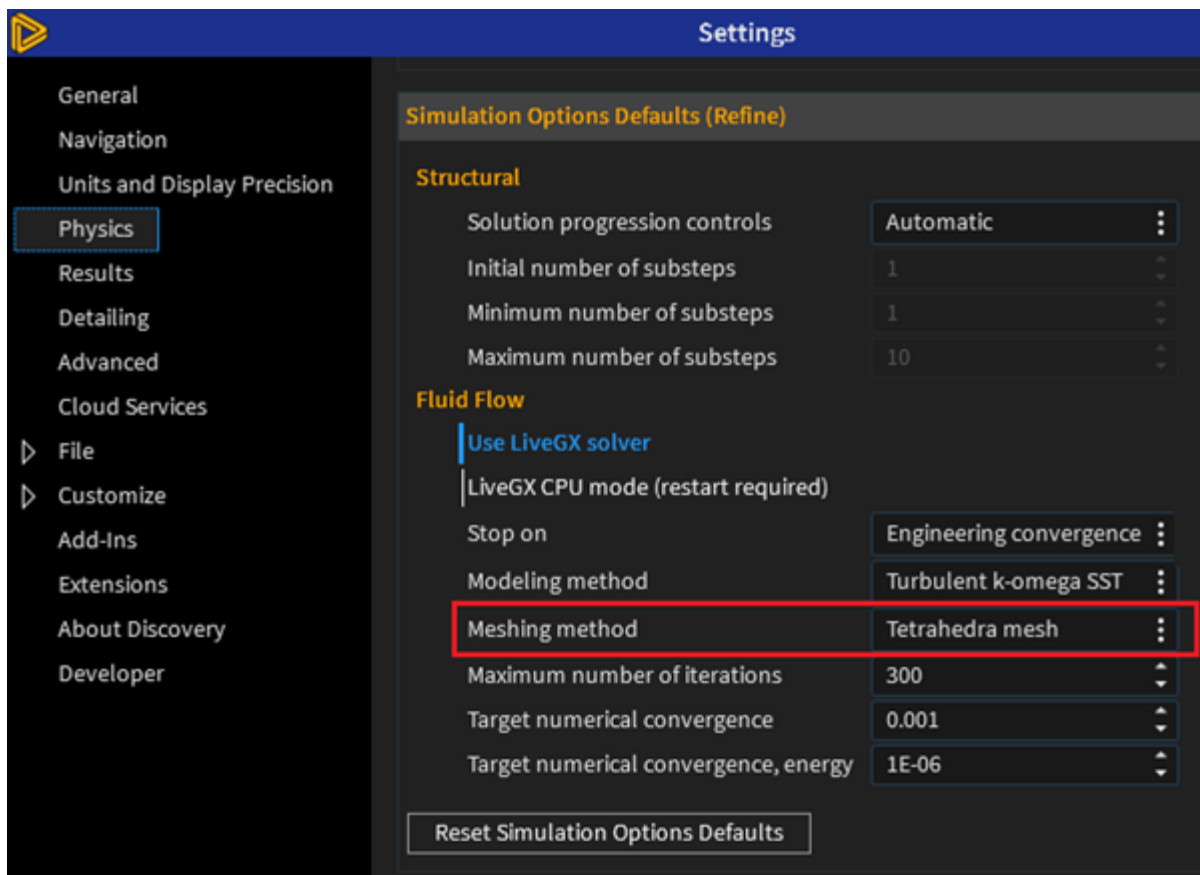
2.1.1. Fluid-Fluid Interfaces

You can use the new fluid-fluid **Interface Review** tool to visualize and inspect the interfaces between different fluids in your simulation. The tool allows you to control the detection distance and visually diagnose the overlapping interfaces or conversely check that the interface does exist where it is expected. To review the fluid-fluid interfaces, you must first go to **File Menu > Settings > Physics** and turn on **Show fluid-to-fluid interfaces**. The interfaces can be reviewed by double-clicking the automatically created condition under **Fluid-Fluid Interfaces** in the physics tree. Alternatively, under **Review** on the **Simulation** tab, select **Default Fluid-Fluid Interfaces** to load the **Interface Review** tool. Each interface pair is shown in the model as a single dot.



2.1.2. Fluid Flow Meshing Method Default

You can now select the meshing method for your simulation under **File Menu > Settings**. Go to **Physics** and under **Simulation Options Defaults** for fluid flow, select the **Meshing method**. The default meshing method is **Tetrahedra mesh**. Depending on the complexity of your model and the solution accuracy you can change the default setting to **Polyhedra mesh**. The meshing method affects the speed and accuracy of the simulation, memory and storage requirements.



2.2. Structural

The following enhancements related to structural simulations are available in 2024 R2.

2.2.1. Structural Solver (Explore)

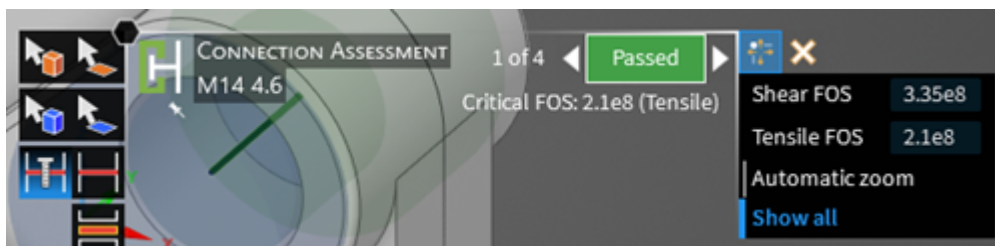
Structural solver enhancements for **Explore** include:

- Improved performance, with solution times up to 3x faster for thin geometries (dependent on GPU hardware)
- Enhanced GPU memory usage, with up to a 50% memory reduction when using a remote mass, a bolted connection, or both (dependent on GPU hardware)
- Improved solution accuracy for all models that include frictionless support or idealized sliding contact by ensuring the correct behavior for planar and cylindrical surfaces using the enhanced surface normal calculation

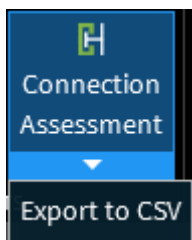
2.2.2. Connection Assessment Tool (Explore)

Connection assessment tool enhancements include:

- The Connection Assessment Tool now determines the bolt factor of safety based on modified AISC ASD criteria, to aid in the simulation of bolted connections.



- You can now save the connection pass/fail information to a comma-separated values (CSV) text file clicking on the Connection Assessment Tool.



2.2.3. Contact and Interface Review

Contact and interface review tool enhancements include:

- The functionality of the contact review and interface review tools was improved by clarifying that contacts and interfaces are **Excluded** rather than **Prevented**.

- **Detection tolerance** for contacts and interfaces was given greater prominence to highlight its priority and increase ease of use.

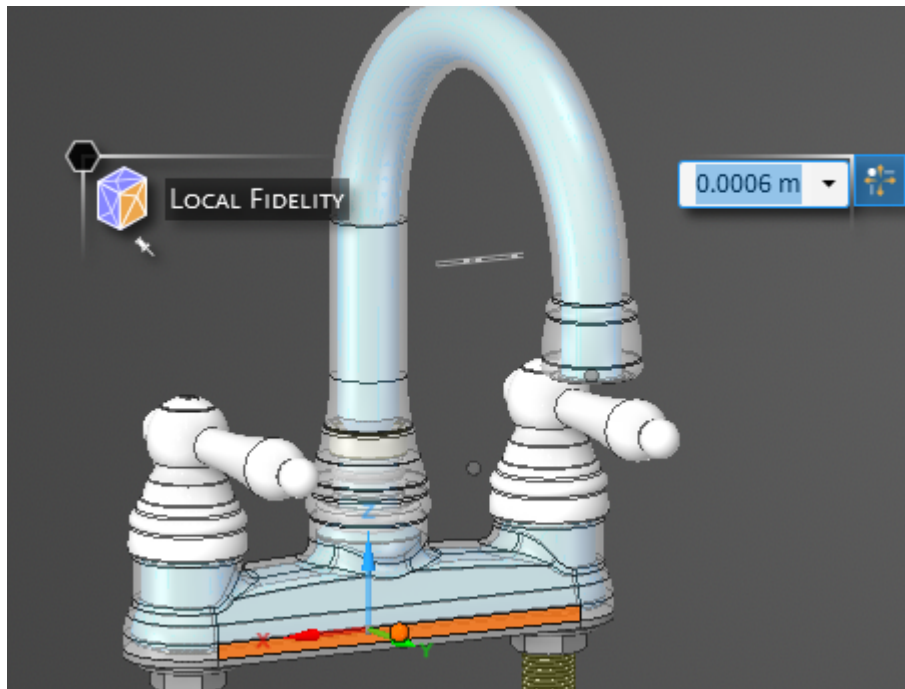


2.3. Meshing

The following enhancements related to meshing are available in 2024 R2.

2.3.1. Local Fidelity in Explore

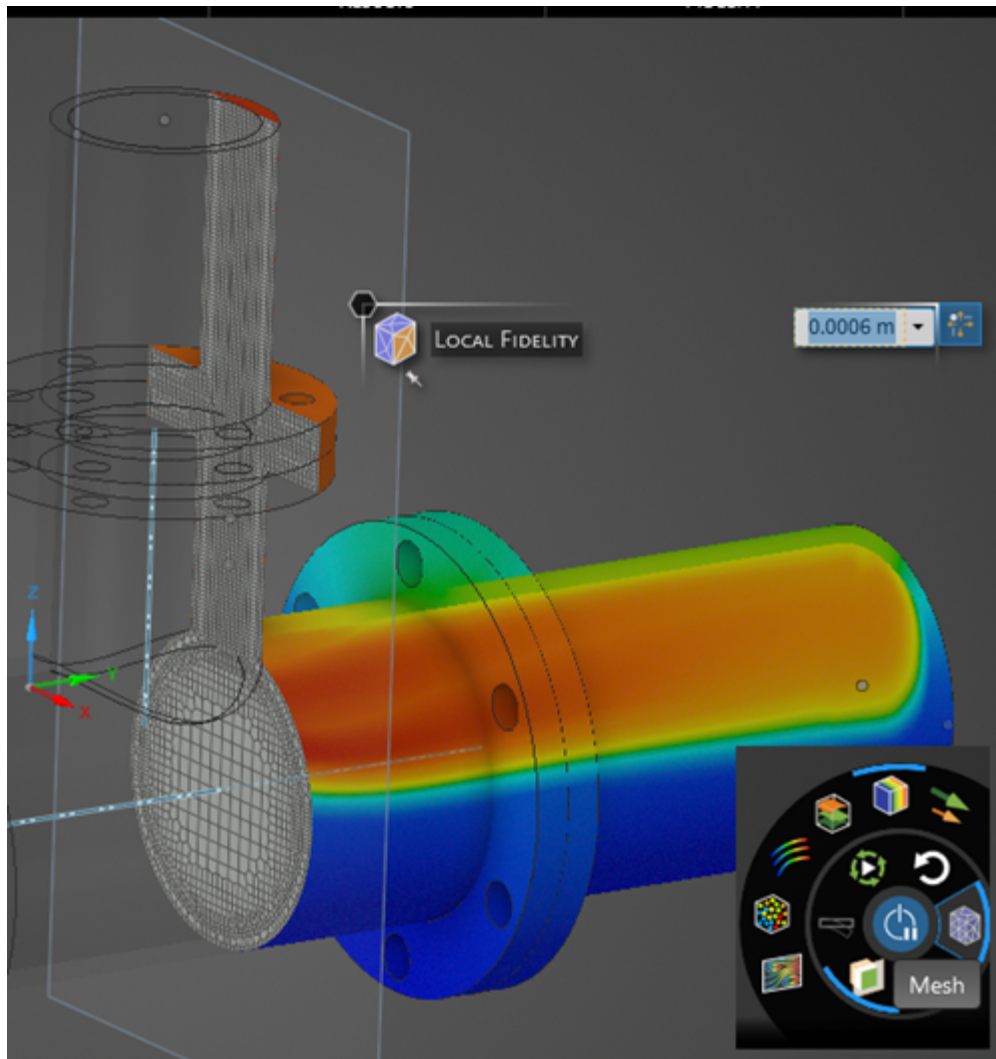
You can now apply local fidelity in **Explore** and **Refine** for fluid flow simulations. Use **Local** on the **Simulation** tab, to manually set lower or higher result resolutions. In **Explore**, for structural and solid thermal simulations, local fidelity can only be assigned to bodies and for fluid flow simulations it can only be assigned to faces. In **Refine**, you can apply local fidelity to bodies, faces, and edges for all physics types.



2.3.2. Mesh Display for Explore Fluids with Local Fidelity

Explore fluids now includes the capability to display the mesh for fluids and fluid-solid (conjugate) heat transfer simulations when a local fidelity is specified. The mesh can be displayed on a cut-plane or the surface mesh can be displayed by using the **Mesh** option in the Results arc. This new

capability improves the ability to review the discretization of the model and effects of local fidelity on the fluids mesh.

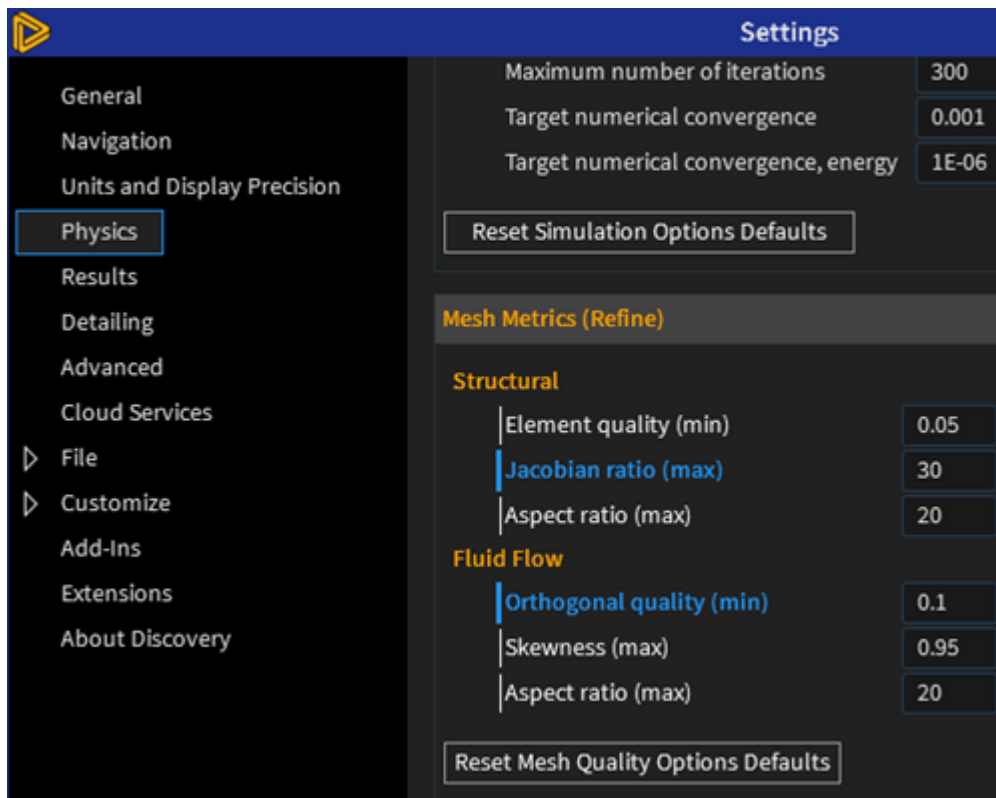


2.3.3. Mesh Quality Metrics

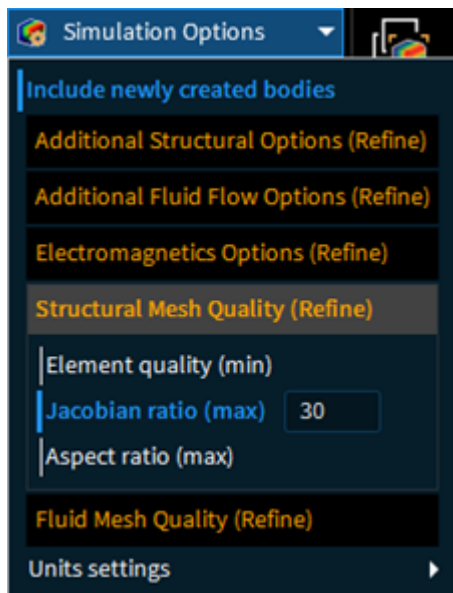
In the **Refine** stage, you can now be notified after simulation of any display elements that exceed quality thresholds. These notifications ease the validation of mesh quality and help you identify areas to apply local fidelity.

- For structures, metrics include minimum element quality, maximum Jacobian ratio, and maximum aspect ratio.
- For fluids, metrics include minimum orthogonal quality, maximum skewness and maximum aspect ratio.

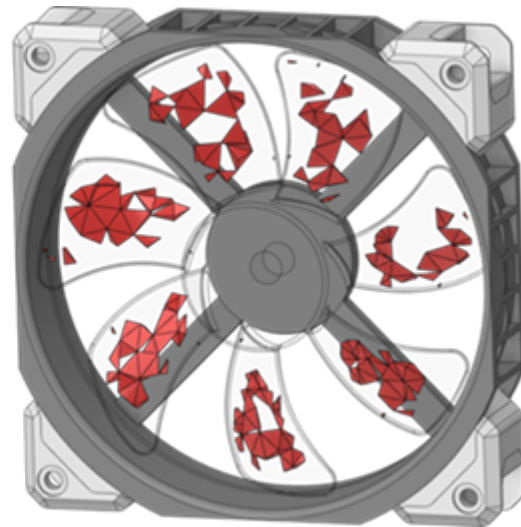
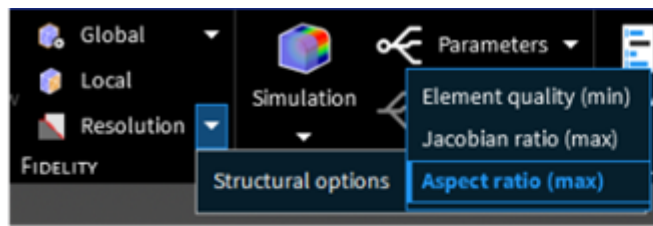
You can define default thresholds for these mesh quality metrics in the **Physics Settings**:



Which can be overridden for any project in the **Simulation Options** on the **Simulation** ribbon.



You can view the elements that do not meet the quality threshold either by clicking the link in the notification message or by selecting the mesh metric you want to view from the **Resolution > Structural options** dropdown list.

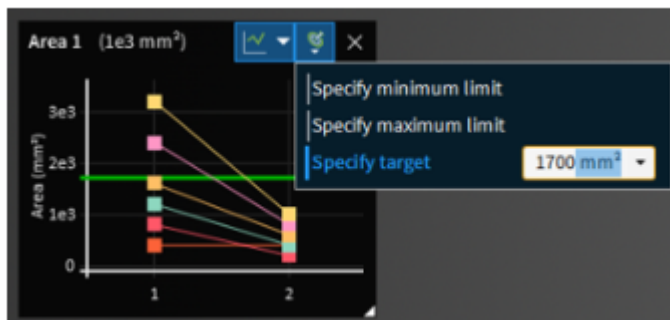


2.4. Results Monitors

The following enhancements related to results monitors are available in 2024 R2.

2.4.1. Output Parameters

You can now specify the target for your monitor.



You can generate a project report showing details about your project. Output parameter monitor data for the current defined variation is displayed in the report.

Solved defined variations are plotted as points on the chart. Hover over a point to see the associated values and the variation they represent. Right-click to access more options such as **Export to CSV**.

2.4.2. Mass Flow-Weighted Average for Monitors

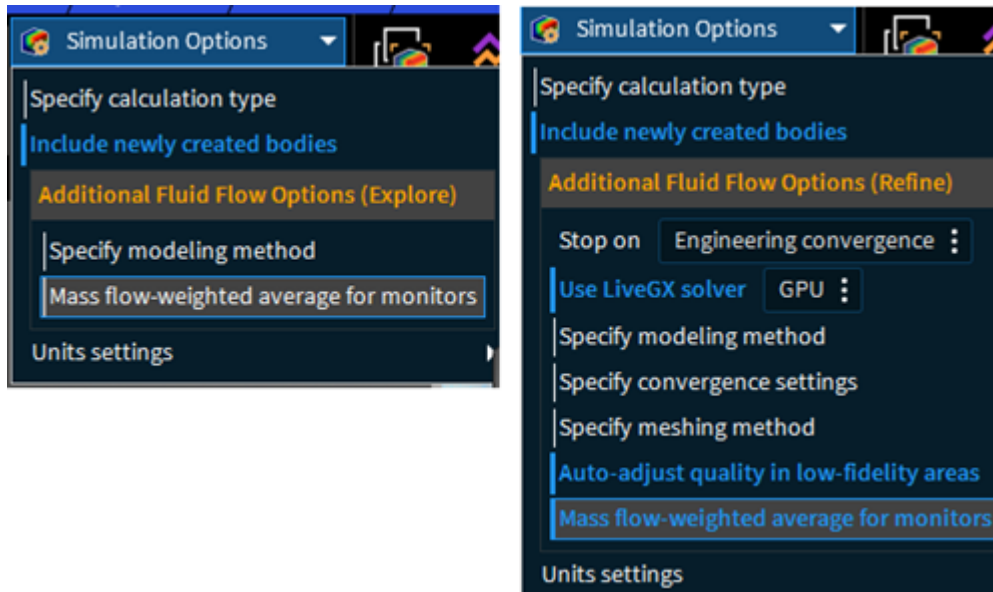
Mass flow-weighted averaging is new in Discovery 2024 R2.

Discovery generally uses area-weighted averaging for all results monitors. You can now use mass flow-weighted averaging instead for the following results variables:

- Temperature
- Temperature uniformity

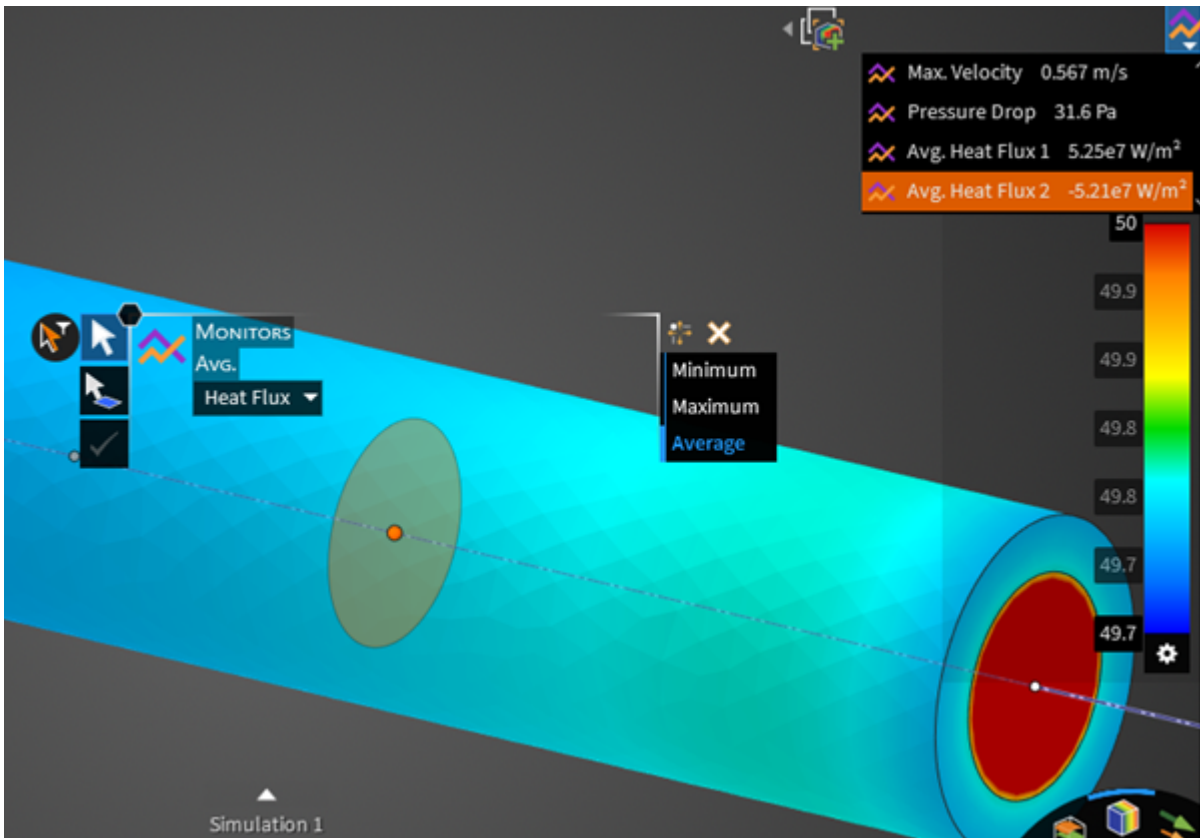
- Total pressure
- Pressure drop

Under **Simulation Options**, on the **Simulation** tab, click **Additional Fluid Flow Options** to see the options in the expanded panel. Click **Mass flow-weighted average for monitors** if you want monitors calculated using mass flow-weighted averaging.



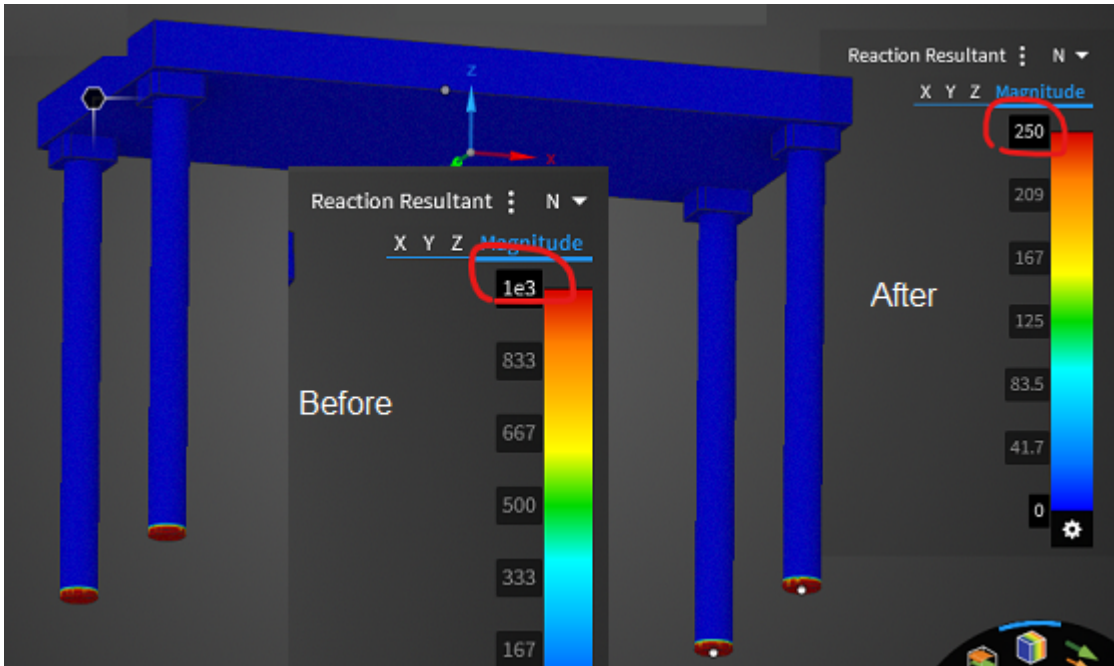
2.4.3. Surface Heat Flux Monitor

You can now monitor the heat flux across a specified surface in **Explore** and **Refine**. On solid thermal bodies, you can monitor the heat flux magnitude or vector components. On faces, the monitor is for heat flux monitor normal to the surface.



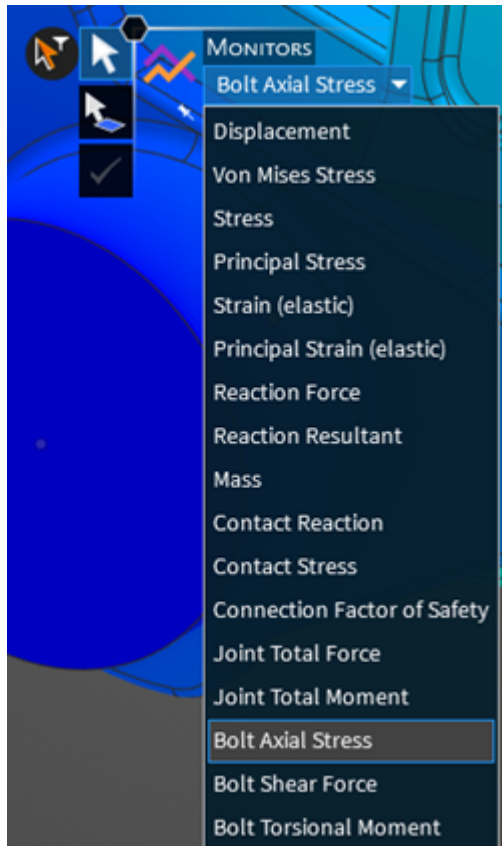
2.4.4. Reaction Resultant Monitors

In previous versions of Discovery, the reaction resultant magnitude was over the entire support surface. Now, in **Explore**, the reaction resultant magnitude is distributed between the applied surfaces. However, in **Refine**, it is still over the entire support surface.



2.4.5. Bolt Monitors (Explore)

In the **Explore** stage, the workflow for upfront simulation of bolted connections is now enhanced to include support for bolt force and moment monitors, including axial stress, shear force, torsional moment, and factor of safety. Used in combination with the Connection assessment tool, these new bolt monitors improve the workflow for defining and evaluating the structural integrity of bolted connections based on a modified AISC allowable stress criterion.

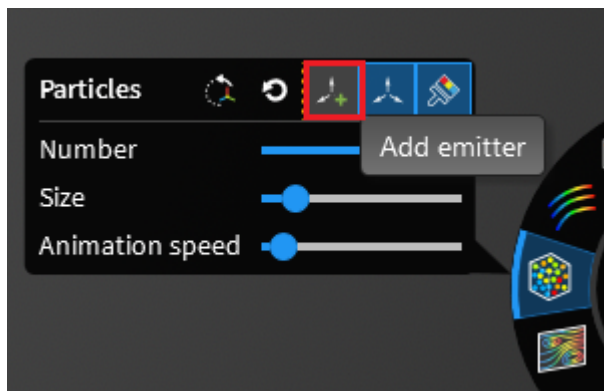
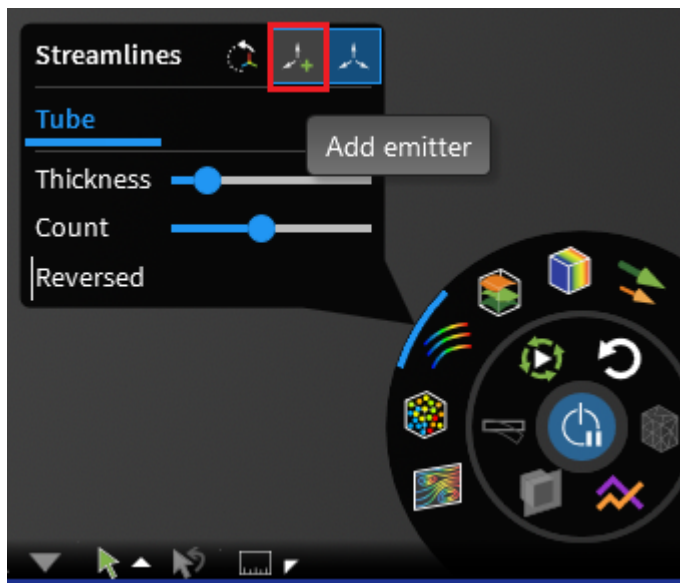


2.5. Results Enhancements

The following results enhancements are available in 2024 R2.

2.5.1. Additional Fluid Emitters

You can now include additional particle and streamline emitters. In the Results arc, turn on **Streamlines** or **Particles** and from the options, select **Add emitter**. The first emitter is added to the first inlet while additional emitters are added to subsequent inlets and finally offset from one another at the last inlet. Additionally, you can place an emitter anywhere on your model by choosing **Add a Streamline (or Particle) Emitter** from the context menu.

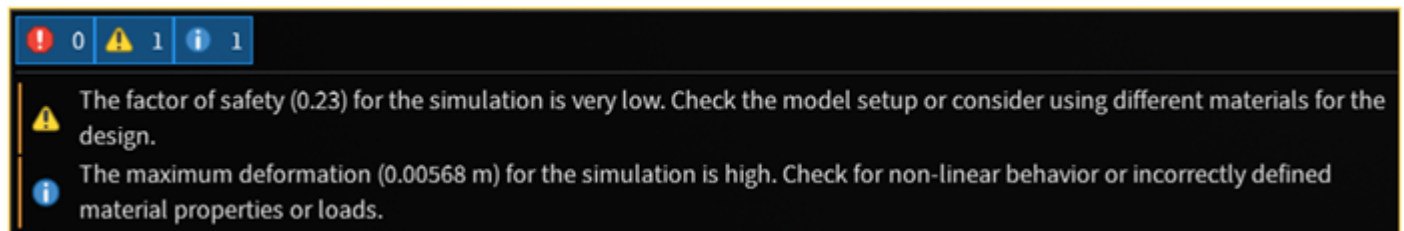


2.5.2. Improvements to Results Display

Displayed iso-surfaces results now update instantly as you interact with your model.

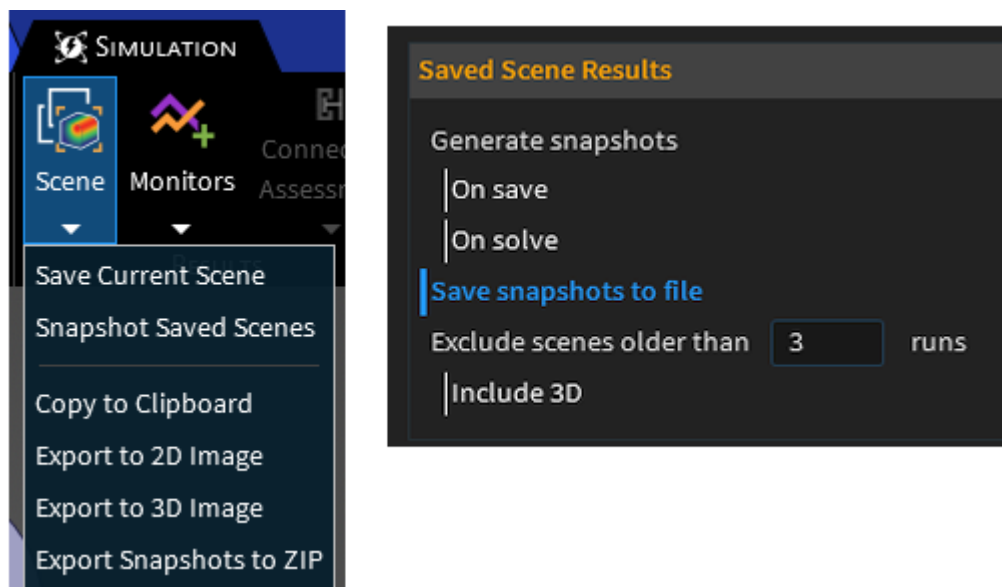
2.5.3. Enhanced Messages for Results Confidence

- Warning or informational messages now highlight questionable results.
- Results are checked for low factor of safety, large deflection, high velocity, and high or low temperature values.



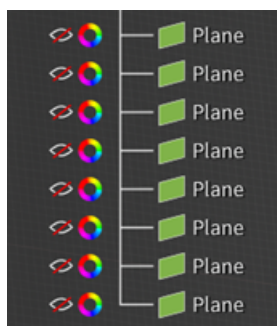
2.5.4. Exporting Saved Scenes Snapshots

Saved scenes snapshots can now be saved and exported to a ZIP archive with the new **Export Snapshots to ZIP** command. Also, the settings panel (Results > Saved Scenes) reflects some reorganization and is easier to use.



2.5.5. Results on Multiple Cut Planes

If you have added cut planes from the Design tab, you can now view results on all those planes by right-clicking any one of those planes in the model tree and selecting **Show results**. To view the results only for that particular plane, right-click and select **Show results on selection only**.



2.6. Hybrid Modeling

Hybrid modeling enhancements include:

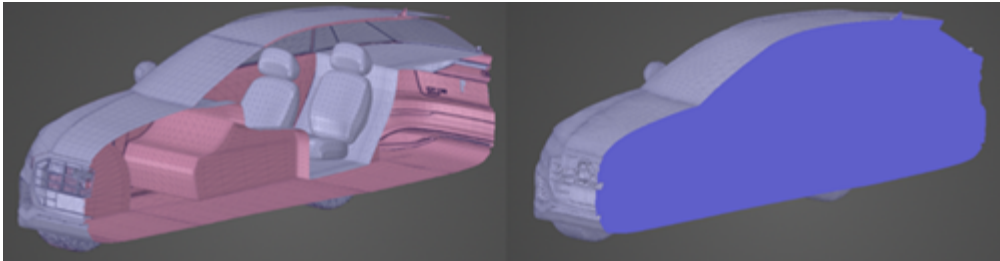
- You can now use the **Auto Skin** tool to create the SubD hybrid model from faceted models with topology optimization results.

The facet regions are converted to a combination of traditional and SubD faces. The SubD hybrid model retains the traditional analytic faces such as planes or cylinders for regions that require precise topology. The remaining facets are converted to SubD faces which allow for flexible editing.

- Once you have created the hybrid model of the resulting optimized shape, you can right-click the hybrid model in the Model tree select the **Create Validation Simulation** option to create a validation simulation that includes all of the physics conditions from the topology optimization simulation applied to the hybrid model.

2.7. Shrinkwrap

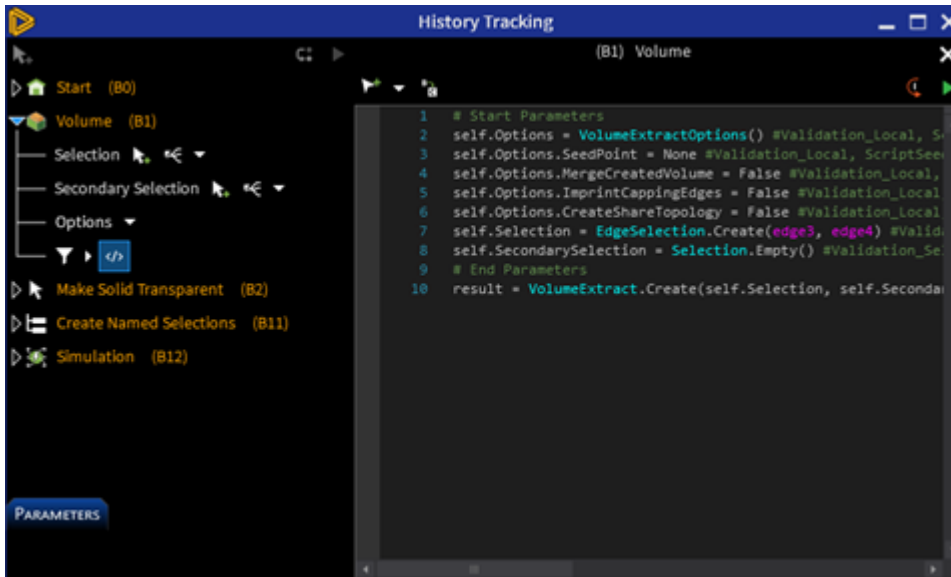
For models with large openings, the new **Remove interior** option enables covering all openings and holes while removing interior voids. Specify the **Fit tolerance** or use a value of zero to automatically compute facet sizes based on the geometry.



2.8. History Tracking

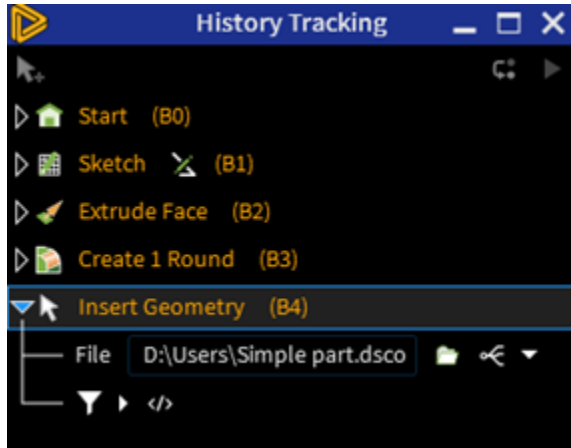
The following enhancements related to history tracking are available in 2024 R2.

History Tracking is now accessible from the **Automate** group in the **Design** tab. The improved window is more compact and also includes the Script Editor for editing the block script. Visual treatment for colors and icons has been improved for better contrast and usability.



- Block selection and expansion has been improved.
- Additional options like the Script Editor or Edit sketch can be accessed without expanding or selecting blocks.
- When you stop tracking history, the window closes. Additional modeling or physics changes will delete the history.

- **File > Insert Geometry** is now supported for History Tracking.



2.9. Modeling Tools

The following enhancements related to modeling tools are available in 2024 R2.

Note:

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

2.9.1. Improvements to Modeling Documentation in Discovery

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

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

2.9.2. Hide and Show Faces

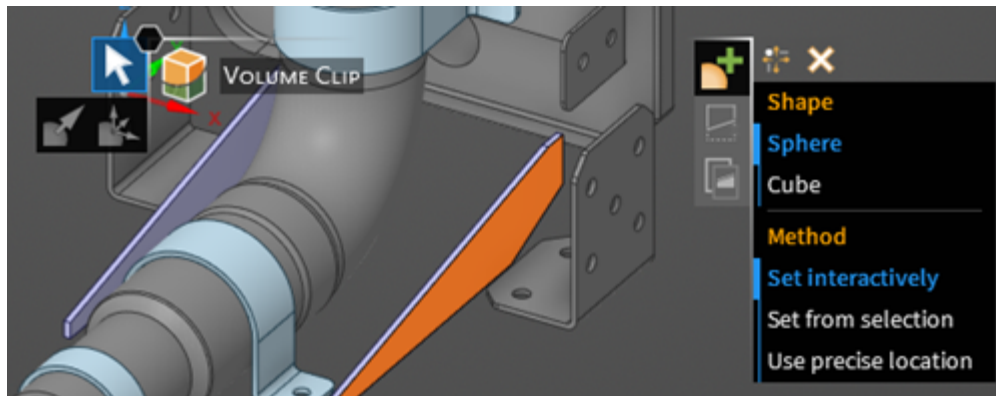
You can now easily hide and show faces by accessing the new Visibility group on the Display ribbon.



2.9.3. Clip Volume

A new advanced Clip Volume tool is now easily accessible from the new Visibility group on the Display ribbon. You now have the option of a spherical or cubic volume and can choose whether

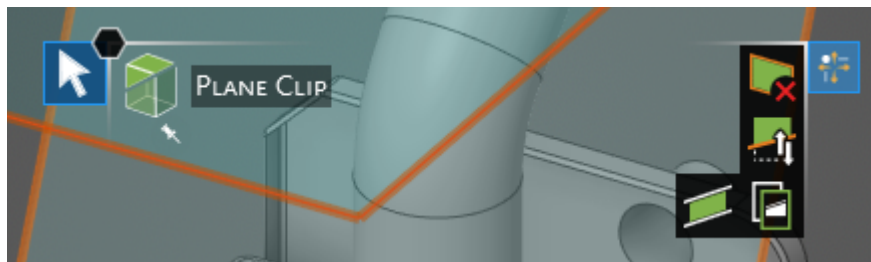
to clip the design interactively or based on a selection. You can also clip the design based on a specified location and size.



2.9.4. Clip Plane

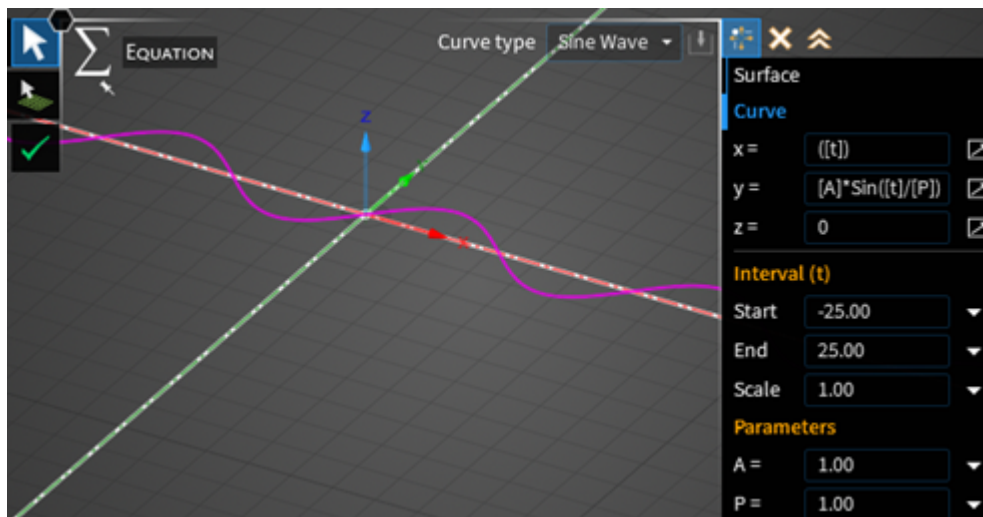
A new advanced Clip Plane tool is now accessible from the new Visibility group on the Display ribbon. This tool enables you to create a clip plane to show geometry on one side of the plane or the other. Once you've created a clip, you can reverse the direction of the clipped view, or merge clips together. Planes, axes, and annotations are not clipped.

You can select up to ten planes to set a clipped volume. Planes selected for clipping display with orange boundaries in the graphics window.



2.9.5. Equation Tool

The **Equation** tool includes options to create curve geometry from preset equations or edit existing curves. You can also create custom curves and save them for further use.



2.9.6. Shared Topology Review

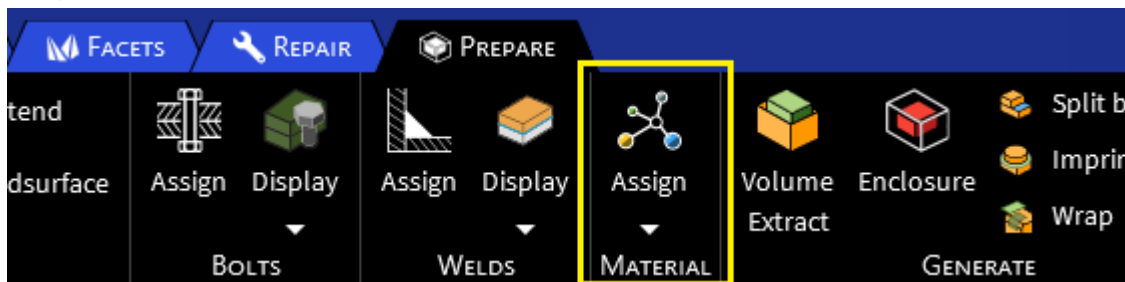
When using the **Shared Topology Review** tool, the new **Keep visible** option enables shared entities to display persistently when using other tools.

2.10. Materials

The following enhancements related to materials are available in Discovery 2024 R2.

2.10.1. Availability of Materials Tools

Materials tools are now available on the **Prepare** tab, where Discovery users with Discovery Modeling licenses (disco_level1) can access them.



Previously, Materials tools were available only on the **Simulation** tab for users with Discovery Simulation licenses (disco_level2).

Material import settings are now off by default in the **Settings** panel accessed under **File > Settings > File**.

2.10.2. Materials Data for Simulation

Materials Data for Simulation (MDS) is available through an optional license that provides access to simulation-ready materials data for a broad coverage of materials classes, enabling engineers to create more accurate simulations with trusted data. For easy access, MDS is embedded directly into Discovery and currently contains 771 materials covering a wide range of material classes.

Material Class	Count
Ceramics	47
Composites	37
Fluids	37
Foams	16
Glasses	15
Honeycombs	3
Magnets	27
Metals - ferrous	153
Metals - non-ferrous	226
Polymers	176
Woods	34
Total	771

Developments for 2024 R2 have focused on continuing to provide accurate and reliable simulation-ready materials data with standard updates to point and curve data.

There are no new materials in the 2024 R2 release of MDS, but some data values have been updated.

Sustainability data has been added for 39 materials and it is presented for three key environmental indicators:

- **Climate change (CO₂-eq).** This is the CO₂-equivalent of the production of 1 kg of the material from a combination of its ores and some recycled content. Climate change refers to a change in the state of the climate that can be identified by shifts in the mean and/or the variability of its properties, and that persists for an extended period—typically decades or longer. It is calculated based on global warming potential (GWP), developed by the Intergovernmental Panel on Climate Change. The GWP model allows comparisons of the global warming impacts of different gases, reported as carbon dioxide equivalents (denoted as CO₂-eq).
- **Embodied energy.** This is the embodied energy of the production of 1 kg of the material from a combination of its ores or feedstocks and some recycled content. It represents both the direct and indirect energy use throughout the life cycle of a good or a service, and it is based on the cumulative energy demand method developed by ecoinvent. It includes renewable and non-renewable energy resources (biomass, fossil, geothermal, nuclear, primary forest, solar, water, and wind), and the value reported is the sum of all of these resources.
- **Potential to recycle.** This indicates whether, when the material reaches its end of life, it can be recycled, regardless of the recycle's quality. A material is designated as recyclable if it can be reprocessed, by generally available and economically viable means. It includes processing of both new and old scrap: new scrap is material that is generated during the manufacturing process, and old or post-consumer scrap is that which has been used by a consumer.

For additional sustainability data, see the Ansys Granta Selector and Ansys Granta MI products. These products have technical, economic, and sustainability data on several thousand materials which can be accessed in Discovery as required.

2.10.3. Materials Data for Simulation - Sample Library

The Materials Data for Simulation Sample library provides 39 materials free of charge to Discovery users with basic room temperature point data for mechanical and thermal properties.

Sustainability data has been updated for all the materials in the sample library and it is presented for three key environmental indicators: **Climate change (CO₂-eq)**, **Embodied energy**, and **Potential to recycle**. See [Materials Data for Simulation \(p. 19\)](#) above for details.

2.11. Tree and Advanced Selection Panel

Tree enhancements include:

- Many objects in the UI can be undocked and moved in and out of the Discovery graphics window to allow more room to work.



- Control the division and visibility between the model tree and the physics tree by grabbing and moving the divider line.



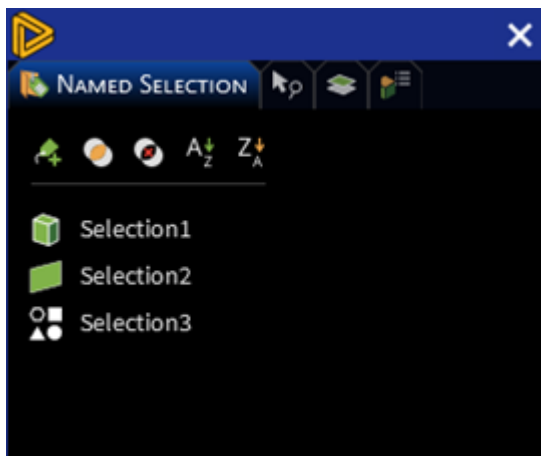
- Images now have a separate node in the Model tree.

Images can be pasted into the model from the clipboard or inserted using **File > Insert Geometry**. In either workflow, there must be an existing plane (face or datum plane) to drop the image onto.



Advanced Selection enhancements include:

The **Named Selection**, **Power Selection**, **Layers**, and **Properties** panels have been combined into a tabbed window that can be moved around the screen or to other monitors.

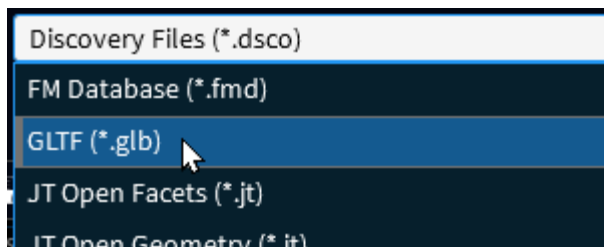


2.12. File Settings

The following enhancements related to file settings are available in Discovery 2024 R2.

2.12.1. GLB Export

Curved vectors can now be exported to GLTF (*.glb) format. In the **Save As** dialog box, select the **File Type** as **GLTF** and click **Save**.

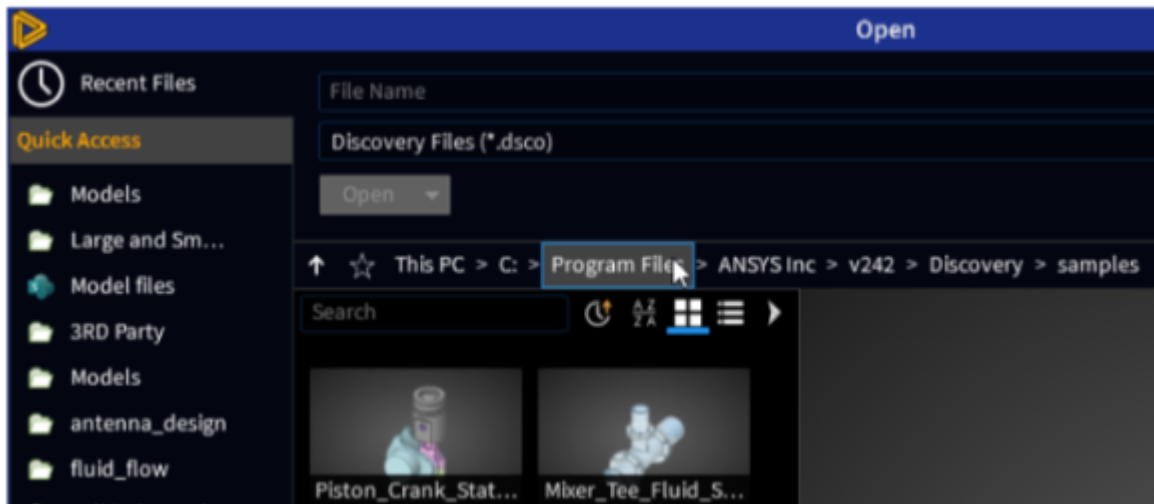


Curved vectors can also be exported by clicking **Scene > Export to 3D Image**. A save file dialog prompts for a file name and location to save.

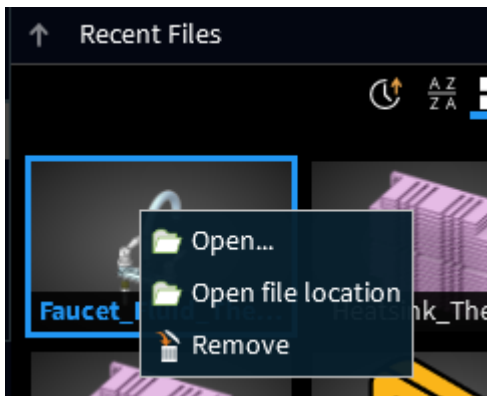
2.12.2. File Options

File options enhancements include:

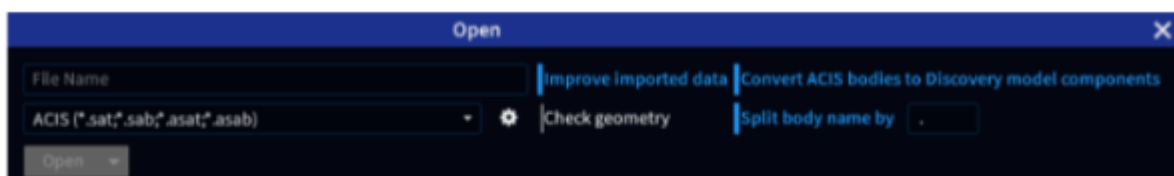
- File details are now shown in the **File Open** dialog boxes and Home Page when using the list view. An improved Breadcrumb for the **File Open** dialog boxes is now available. Locations in the path are now clickable for jumping directly to the location, including cloud locations. If a breadcrumb is too long, scroll arrows appear to let you scroll and see the hidden parts of the path. You can also scroll with the mouse wheel.



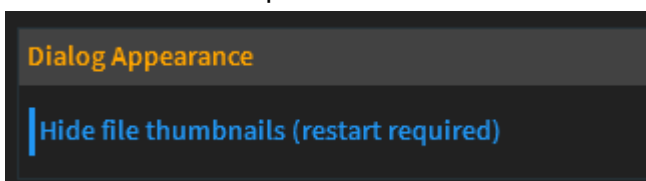
- A new **Open file location** button has been added to the context menu for Recent files. This helps to navigate directly to the recent file's directory.



- File Import settings are now shown in the **File > Open** dialog box by default.

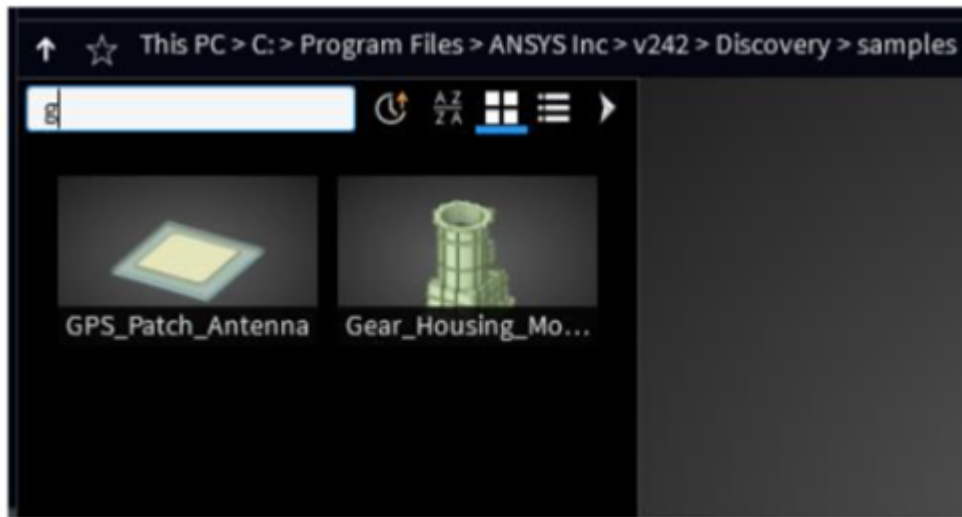


- An option has been added to **Settings > File > General** to hide detailed thumbnails in the File options and the Home Page. The option is OFF by default. Only the Discovery logo thumbnails are used when the option is ON.



- Additional file details are shown in the File options and Home Page. **Recent Files** shows when the file was last opened, the path, and the file size. **Sites** shows who last saved the file and when, and the file size. **Shared With Me** shows who shared the file and when, and the file size.

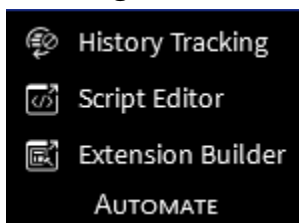
- Search has been updated in the file **Open** and **Save As** dialog boxes. After you start typing and then stop the tiles update after a brief delay. After typing a letter, stop typing briefly to see the tiles update.



2.13. User Interface Enhancements

User interface enhancements include:

- More tips are available to be shown randomly at startup.
- Transparency and color are now retained when the volume is updated after a Pull operation.
- **History Tracking**, **Script Editor**, and **Extension Builder** are available in an **Automate** group in the **Design** tab.



History Tracking can no longer be accessed from the Model tree.

2.14. Extension Builder Enhancements

The following user settings enhancements are available in Discovery 2024 R2.

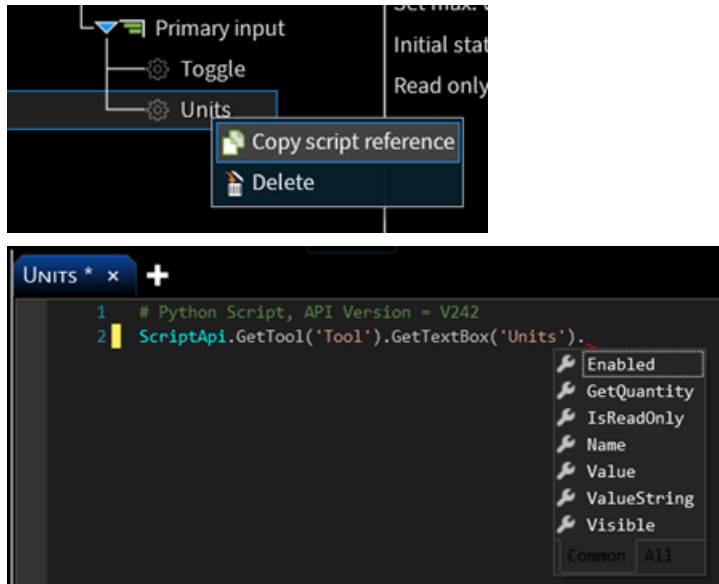
2.14.1. Copy Script Reference and Close All Script Tabs Button

To assist with script writing, right-click a **Tool** or **Tool Option** in the **Extension Tree** and choose **Copy script reference** to copy it to your clipboard. You can then paste the relevant script reference into the **Script Editor**.

Additionally, a new



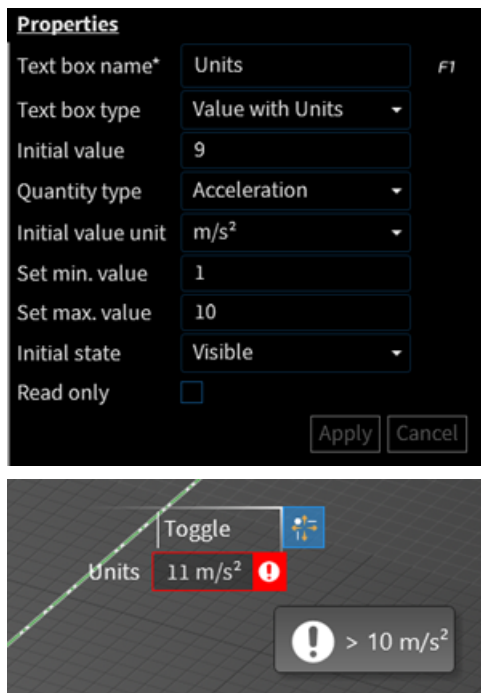
button has been added at the top of the **Script Editor** panel to close all script tabs, leaving only active scripts and those associated with highlighted tree items open.



2.14.2. Text Box Enhancements

We've introduced a new **Read only** option for text boxes, allowing you to include text boxes with fixed values that cannot be edited.

Additionally, the **Value with Units** text box type lets you define an initial value, along with minimum and maximum thresholds. After previewing or publishing your extension, any values entered outside of the specified range will prompt an error message to your users.



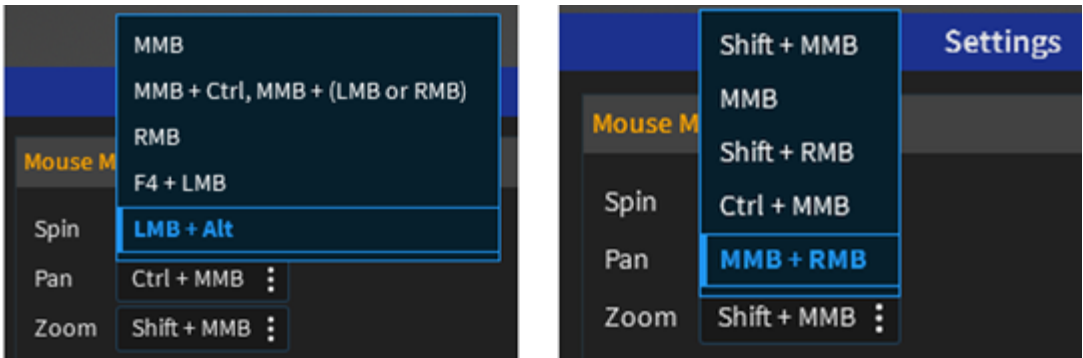
2.15. User Settings Summary

The following user settings enhancements are available in Discovery 2024 R2.

2.15.1. Mouse Settings

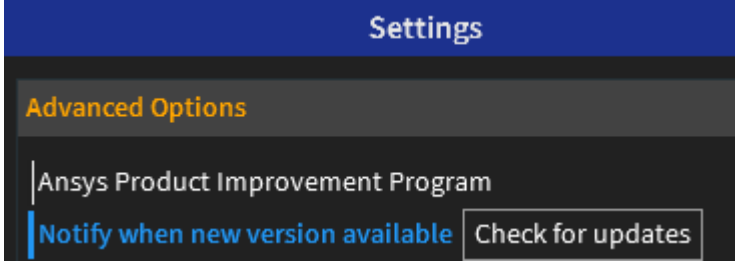
New mouse mappings have been added for **Spin** and **Pan** under **Settings > Navigation**.

LMB + Alt has been added for **Spin** and **MMB + RMB** has been added for **Pan**.



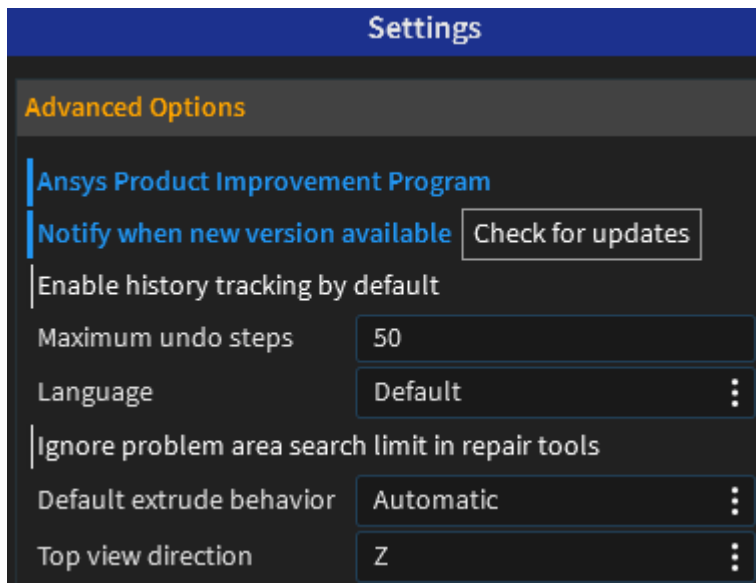
2.15.2. Check for Updates

Select **Check for updates** to be notified when a new version or service pack is available. You can check manually at any time.



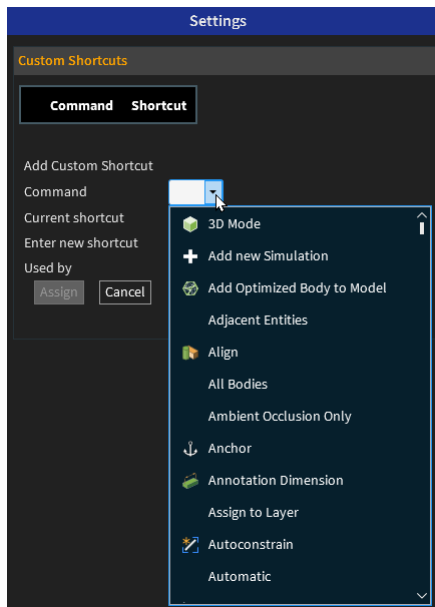
2.15.3. Top View Direction

Set the default top direction of a Discovery session. **Z** is the up axis and **Y** is the down axis.



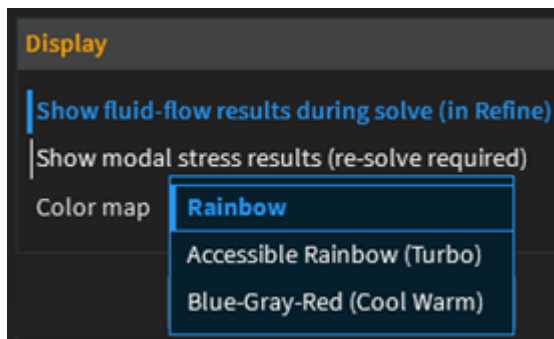
2.15.4. Custom Shortcuts

Creating custom shortcuts is easier now with over 200 commands available in the dropdown to make shortcut assignments.



2.15.5. Improved Color Maps

New and updated color maps address color blindness ambiguity. Color maps now available in **Display** under **Settings > Results** include: standard Rainbow, Accessible Rainbow (Turbo), and Blue-Gray-Red (Cool Warm).



2.16. Performance

The following performance enhancements are available in Discovery 2024 R2.

2.16.1. Responsiveness Improvements

Discovery Release 2024 R2 continues improvement in application performance and modeling responsiveness. You may notice performance improvements in all stages from refresh rates to solution rendering, and improved stability for long running sessions.

Some specific items worth noting include performance improvements in modeling, migrating and opening projects from previous releases, and background physics automation. Progress dialogs improve feedback and demonstrate enhanced responsiveness.

2.16.2. Background Automation for Physics

Some computationally-heavy physics operations may lead to a performance lag in Discovery's responsiveness. **Background automation for physics** (in the **Physics** tab of the **Settings** panel) is now ON by default. With this option, Discovery remains responsive during contact detection and updates the model state asynchronously. This means you can continue editing the model in the Model stage during an update.

2.16.3. Display Support

Display improvements for multiple windows, resolution and scaling, support moving and undocking objects from the main graphics window to display outside of the main Discovery window.

2.17. Installation and Licensing

The following enhancements related to installation and licensing are available in Discovery 2024 R2.

2.17.1. Installation Memory Requirement

The documented memory requirement for installing Discovery has been changed to 16 GB RAM or greater. Previously, it was documented as 32 GB RAM or greater.

2.17.2. HPC Requirement for LiveGX in CPU Mode

Running LiveGX in CPU mode is no longer limited to 4 cores. If more than 4 cores are needed, CPU mode now allows Discovery to acquire the appropriate HPC licenses so you can leverage the full processor.

2.17.3. Command Line Option for -geometryservice Packet

When performing a silent install, include the `-geometryservice` command line option to install the headless geometry service packet. PyAnsysGeometry can use the headless geometry service to execute Python scripts for PyAnsys workflows.

You may install the geometry service packet with or without other products. For example:

```
setup.exe -discovery -geometryservice  
  
setup.exe -geometryservice
```

Installation of the geometry service packet is available only via the `setup.exe` command line option.

2.17.4. Installation Prerequisite Checking

Discovery 2024 R2 checks whether the required prerequisites are present, and it reports the results in several ways:

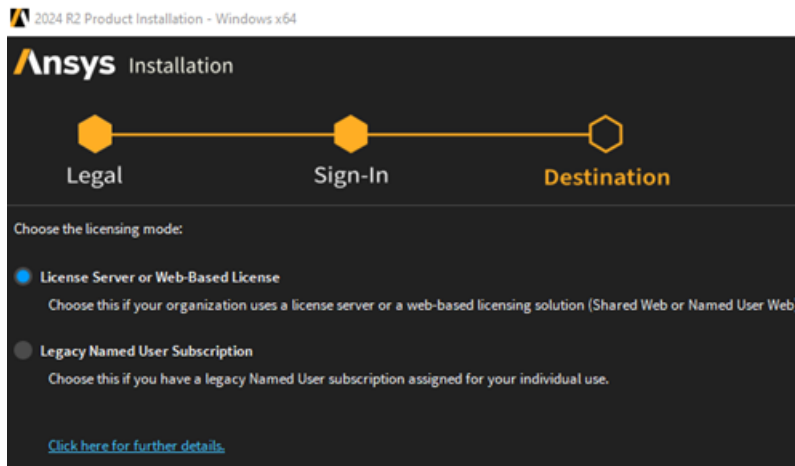
- When you launch Discovery, a dialog box reports any missing or out-of-date prerequisites that were identified.
- Results of the prerequisite checks are captured in the Discovery application log.
- The SID reports any missing or out-of-date prerequisites that were identified when Discovery was launched, as well as results concerning prerequisite checking specific to the **Explore** and **Refine** stages.

2.17.5. Web Named User Licensing

Web licensing now includes new Named User products for Discovery Modeling and Discovery Simulation.

The licensing modes for Discovery have changed as shown in the installation panel below. The panel provides access to detailed information to help you choose the correct licensing mode.

2024 R2 will be the last release to support the **Legacy Named User Subscription** option. It will be replaced with web-based Named User licensing at 2025 R1.



2.18. Online Help Enhancements

Discovery Online Help has a new section focused on all the ways that Discovery provides you with the necessary information and assistance to effectively use the product. Help comes in various forms and formats, including in-product guidance, HTML and PDF, and this section summarizes it all.

2.19. Ansys Discovery Technology Showcase: Example Problems

New at 2024 R2 is the *Ansys Discovery Technology Showcase: Example Problems* manual. This manual presents real-world example problems to showcase the broad capabilities of Discovery.

The following example problems appear in the manual at 2024 R2:

- Natural Frequency Analysis of a Printed Circuit Board
- Transformer and Cold Plate Fluid-Solid-Thermal Heat Transfer
- GE Engine Bracket Topology Optimization
- Butterfly Valve Flow Coefficient

You can download the project files that are used in the example problems directly from the Ansys Help site.

2.20. Ansys 3D Design Verification Manual

For 2024 R2, the *Ansys 3D Design Verification Manual* provides an improved user experience, including:

- Tables summarizing all Discovery test cases, so you can more easily identify and navigate to test cases of interest
- Use of visual cues to distinguish solver and results information in Discovery's **Explore** stage versus **Refine** stage

In addition, the following test cases have been added:

- Structural

- Stress Concentration in a Shaft with a Shoulder Fillet
- Sphere Under Internal Pressure
- Shaft Under Torque Load
- Thermal
 - Thermal Stress in a Bar with Temperature Dependent Thermal Conductivity
 - Deflection of Bimetallic Cantilever Under Thermal Load
- Fluid
 - Laminar Flow Over a Sphere
- Modeling Prepare
 - Midsurface, Extend, Imprint, and Weld
- Modeling Repair
 - Simplify, Small Faces, Merge Faces
 - Split Edges, Extra Edges, and Duplicates
- NAFEMS Benchmarks
 - BMTTA - Benchmark Tests for Thermal Analysis - Volume 2, Cameron, A.D., Casy, J.A.
 - BMTTA, No. 9(i) - 3D Plate with Prescribed Temperature
 - BMTTA, No. 15(i) - 3D Cylinder with Prescribed Heat Flux
 - BMTTA, No. 15(iii) - Hollow Cylinder with Central Heat Source
 - LSB1 - Linear Static Benchmarks - Volume 1, D. Hitchings, A. Kamoulakos & G.A.O. Davies
 - LSB1, No. IC 19 - Thin Cylindrical Solid with Edge Moment
 - LSB1, No. IC 29 - 3D Z-Section Cantilevered Plate
 - LSB1, No. IC 39 - Thin Cylinder and Half-Sphere Solid Pressure Vessel

3. Discovery Caveats and Known Issues

At the time of publication, caveats and known issues in Release 2024 R2 of Discovery include those listed below. Also refer to the *Ansys, Inc. Known Issues and Limitations* document.

- Some non-Discovery project files (such as geometry file formats) split data over multiple files. When synching such local project files to the cloud, only one file is uploaded to the cloud and any additional files associated with the project may not be uploaded. **Workaround:** There is no workaround. This behavior is a current limitation in the cloud. [981316]

- When solving in **Explore**, fidelity settings may cause the GPU to run out of memory and start using the CPU resulting in slow solve times. **Workarounds:** Adjust fidelity slider to a lower position. Disabling shared memory may also be a workaround (see https://nvidia.custhelp.com/app/answers/detail/a_id/5490). [984450]
- When replacing a file (containing comments) with a new file (without comments) using the same name, the comments from the original file may remain. **Workaround:** Use a different file name. [1010914]
- When working with defined variations and reopening a project that was created in a different language, there may be remnant artifacts that remain in the original language. **Workaround:** Rename the parameter in the Parameters table. [1030722]
- When a fluid-solid heat transfer simulation is solved in **Refine** with thermal conductance set on the default contact group, subsequent solves will retain the conductance effect after thermal conductance is turned off. **Workaround:** Create your own contact or contact group rather than relying on the default. OR if the simulation has been solved in **Refine** with thermal conductance set on the default contact group, create new user-defined contacts or contact groups on those pairs from which you want to remove thermal conductance before solving again. This can be done for all contact pairs. [1031334]

SpaceClaim Release Notes

Release 2024 R2 of SpaceClaim contains many new features and enhancements.

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

1. SpaceClaim Product Advisories

SpaceClaim users should be aware of the advisories described here.

SpaceClaim

- SpaceClaim 2024 R2 will be the last version based on ACIS. You should plan to migrate your geometry preparation workflows to Discovery.
- SpaceClaim will be removed from the Ansys unified installers at Release 2025 R1.

API Version Support

API versions prior to V22 will be deprecated at Release 2025 R1.

If you have customized add-ins or scripts, make sure they are upgraded to V22 or later.

2. SpaceClaim Enhancements Overview

Areas where you will find changes and new capabilities in SpaceClaim include:

- [2.1. Scripting](#)
- [2.2. Import Support](#)
- [2.3. Meshing](#)

2.1. Scripting

- The current Scripting version is v242.
- API versions prior to V22 will be deprecated at Release 2025 R1.

If you have customized add-ins or scripts, make sure they are upgraded to V22 or later.

2.2. Import Support

The following Reader versions are now supported:

- CATIA V5-6 R2024
- NX 2312
- SolidEdge 2024
- SOLIDWORKS 2024

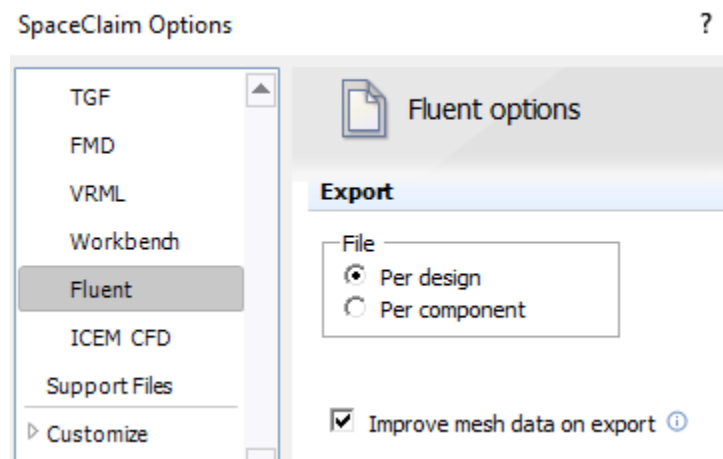
2.3. Meshing

The improvements to the SpaceClaim interactive meshing tools ("Meshing") in this release include:

- Improved mesh export for Fluent and LS-DYNA.

Fluent Mesh Export

The **Improve mesh data on export** option allows you to merge Fluent zones during export.



Scripting support has also been added for the **Improve mesh data on export** option.

LS Dyna Export

The LS-DYNA output interface has been enhanced to support larger meshes by now using 10 digits for node/element numbers.

- Robustness and Performance improvements

3. SpaceClaim Caveats and Known Issues

At the time of publication, caveats and known issues in Release 2024 R2 of SpaceClaim include those listed below. Also refer to the *Ansys, Inc. Known Issues and Limitations* document.

- When using **Block Recoding** to track the import of CAD geometry, **Import Options** are only available in the start block after a **File > Open**. **Import Options** are not available from the **File > Insert** blocks. [1001690]

DesignModeler Release Notes

1. DesignModeler Product Advisories

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

2. DesignModeler Enhancements

DesignModeler 2024 R2 has no new features or enhancements.

3. DesignModeler Caveats and Known Issues

At the time of publication, there are no caveats or known issues in Release 2024 R2 of DesignModeler. Also refer to the *Ansys, Inc. Known Issues and Limitations* document.

Ansys Structural Products

Licensing

At Release 2024 R2, Ansys Mechanical products have transitioned to the new Ansys Common Licensing model. Additional license increments have been added. The previous bundled license increments that enable multiple applications (for example, ansys, mech2, mech1) remain unchanged. For more information, see [Summary of New and Changed Features in the Ansys, Inc. License Management Guide](#).

Release notes are available for the following Ansys Structural products:

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

[Mechanical APDL \(p. 55\)](#)

[Autodyn \(p. 73\)](#)

[Aqwa \(p. 75\)](#)

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

[Material Designer \(p. 81\)](#)

[Additive Suite \(p. 83\)](#)

[Sherlock \(p. 85\)](#)

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

[Motion \(p. 93\)](#)

Mechanical Application Release Notes

This release of the Mechanical application 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. General
6. Graphics
7. Geometry
8. External Model
9. Contact and Connections
10. Mesh
11. Additive Manufacturing Process Simulation
12. Acoustics Analyses
13. Coupled Field Analyses
14. Analysis
15. Fracture
16. Structural Optimization
17. Motion
18. Loads/Supports/Conditions
19. Imported Loading Conditions
20. Solution
21. Explicit Dynamics
22. Results

1. Changes in Product Behavior

Release 2024 R2 includes new features and enhancements that result in product behaviors that differ from previous releases. These behavior changes are presented below.

- **HPC Platform Services (HPS)**. The Ansys **Distributed Compute Services (DCS)** application has evolved into a newer family of technology components represented by **HPC Platform Services (HPS)** while the former DCS framework has been completely removed. HPS supports the same set of functionalities as DCS when used from Mechanical.

- **Fracture.**
 - For all analytical cracks as well as the **Arbitrary Crack**, the default setting of the **Crack Faces Nodes** property has changed from **Off** to **On**. The application generates the associated crack face Named Selections by default.
 - If cracks are included in the analysis, and you add a new **Pressure** load, the **Applied By** property is set to **Direct** by default. In prior releases, the default setting was **Surface Effect**.
 - Now, when you define **Pressure** and **Force** loads using tabular data, they no longer support the **Independent Variable** property when it is set to **Step**, if a **SMART Crack Growth** object is also defined.
 - During a SMART Fatigue Crack Growth analysis, the MAPDL commands written to the input file for Increment of Crack Extension options have changed. Now, the command **CGROW,SOPT,DAMN/DAMX** is used instead of **CGROW,FCG,DAMN/DAMX**.
- **Contact Split Program Controlled Behavior Change.** When the **Contact Split (DMP)** property (**Analysis Settings > Advanced**) is set to **Program Controlled**, the application now automatically performs contact pair splitting during the solution when the number of contact regions is less than the number of:

- Cores when the **Distributed** option is enabled.

Or...

- Processes when the **Hybrid Parallel (Mechanical APDL)** option is **also** enabled.

In prior releases the application always performed contact split when **Contact Split (DMP)** property was set to **Program Controlled**.

- **Program Controlled Setting of the Constraint Type Property.** For **Bonded** contact using the **MPC** formulation, if the **Program Controlled** setting of the **Constraint Type** property does not select either **Projected, Displacement Only** or **Distributed, All Directions**, then automatic constraint type detection is performed internally by the solver. And, see the description for KEYOPT(5) in the **TARGE170** element section of the *Mechanical APDL Element Reference* for a listing of the internal settings.
- **Rigid-To-Flexible Contact.** In the previous release, for a Contact Region defined between a rigid body surface and flexible body, the application created a duplicate layer of target elements (called an insulation layer). This resulted in an overlapping of the target elements at the same location and caused a performance bottleneck for large assembly models. Now, this duplicate layer of target elements is no longer created and therefore improves solution processing performance for those models.
- **Contact Properties** in Motion have been updated. Refer to ??? for details and see the table below for migration paths for affected settings. Note that some **Program Controlled** default options have been changed. Based on these changes, results may differ from solutions in 2024 R1.

Settings in 2024 R1		Migrate to these settings for 2024 R2	
Surface Type = Surface	Node Only Option = Use	Surface Type = Multi Point	Node Only Option = No

Settings in 2024 R1		Migrate to these settings for 2024 R2	
			Longer Supported
Stiffness Type = d-P Spline	Damping Type = Program Controlled (Constant)	Damping Type = Program Controlled (Ratio)	-
Stiffness Type = d-P Spline	Point Check = Program Controlled (Both)	Point Check = Program Controlled (Target Points Only)	-

- **Bushing Joint** . Now, when you set the **Formulation** property to **Bushing**, the default setting of the **Element Coordinate System** property has changed from **Global Coordinate System** to **Reference Coordinate System**.
- **Error Applying Boundary Conditions to Cylindrical Surfaces**. In previous releases, when you applied the boundary conditions listed below on a cylindrical surface, the application did not always apply constraints on nodes whose nodal coordinate system aligned with the Global Coordinate System. The application now processes constraints correctly for these boundary conditions.
 - Frictionless Support
 - Cylindrical Support
 - Displacement (when the **Defined By** property is set to **Normal To**)
- **Create Velocities and Sync Analysis Settings**. In previous releases, the context (right-click) menu option, **Create Velocities and Sync Analysis Settings**, for the **Imported Loads (Group)** object, created individual **Imported Velocity** child objects for each **RPM** value included in the linked upstream system and automatically set the **Mapped Data** property to **To Input File** and synchronized the **Analysis Settings** object with the settings from the upstream result file.

Now, this option creates only one Imported Velocity child object and automatically sets the **Mapped Data** property to **To Binary File**, the **Specify RPM** property set to **All**, and synchronizes the **Analysis Settings** object from the upstream result file.

- **Resource Prediction**.
 - This feature is now only supported with the **Structures AI+** license.
 - In previous releases, the application performed prediction and displayed table data based on a default core count of **4** cores. Now, the application uses the core count specified in the **Solve Process Settings**.
 - Now, to view prediction data from the **Solution Information** object, you must first activate a preference. From the **Options** dialog, activate the **Resource Prediction During Solve** preference in the **Analysis Settings and Solution** group. The preference is available under the **Resource Prediction (Static Structural, Modal and Harmonic Response)** category.

- **Incident Wave Source.** Now, when you select the **Back Enclosed Loudspeaker** option for the **Wave Type** property, the **Pressure** setting for the **Excitation Type** property is no longer supported.
- **Remote Boundary Conditions.** The contact pairs created for **Remote Force**, **Remote Displacement**, and **Moment** that 1) have the **Behavior** property set to **Rigid** and 2) that are scoped to an edge with noncollinear nodes, now use the contact element [CONTA177](#). In previous releases these contact pairs used contact element [CONTA175](#).
- **Point Mass.** In previous releases, the **Element APDL Name** property enabled you to assign an APDL parameter name to the mass element type. Now, the **Element APDL Name** property will be assigned to the mass element number.
- The **Drivetrain Toolkit** in Motion now contains specific parameters for gear Web Mass Information, to allow the use of more accurate mass and moment of inertia values.
- The **Record Frequency Response** option for a Motion analysis in Mechanical **Analysis Settings** has been removed in version 2024 R2. Models using this option will no longer produce accurate results. FFT-related properties have been moved to **Harmonic Acoustics** analysis and you should now access them from there.
- **Tables.** The user interface for creating and editing tables has the following improvements:
 - You no longer need to add a **Tables** object before adding a **Table**. Mechanical now automatically creates the Tables object.
 - You can now add or import tables directly from the **Model > Insert** menu.
 - The icons on the ribbon for importing tables from files and creating tables were updated.For additional table-related changes, see [Loads/Supports/Conditions \(p. 49\)](#)
- **Velocity and Acceleration Unit System Change.** Now, during **Response Spectrum** and **Random Vibration** analyses, acceleration and velocity results use their actual unit systems. In previous releases, these boundary conditions used length as their unit system.

2. Documentation

The following documentation changes and/or enhancements were made for Release 2024 R2:

- [Boundary Conditions in the Mechanical User's Guide](#) underwent significant changes, including:
 - Updates for new product features and user interface changes.
 - A new introduction that describes the workflow for applying boundary conditions in the Mechanical application.
 - A section on stepped and ramped loads (moved from the General Analysis Workflow chapter).
 - Re-organized content that follows the workflow for applying boundary conditions.
- **Structural Optimization Documentation.** New theoretical background documentation has been added to the Structural Optimization guide to describe the [Shape Processing Workflow](#) for the

various methods, such as topology, shape optimization, etc. In addition, theoretical material has also been introduced for [Manufacturing Constraints](#), in particular, for the Member Size feature.

The addition of theoretical documentation will continue throughout future releases and documentation updates.

- New Technology Showcase Chapter: [Chapter 53: Acoustic Analysis of a MEMS Microphone](#)

The acoustic analysis of a MEMS microphone has been added to the Mechanical Technology Showcase: Example Problems.

3. Performance

The following performance enhancements were made for Release 2024 R2:

- **Imported Remote Loads.** The deletion process used when you reimport remote loads from an upstream system has been refactored and the processing time significantly improved.
- **Importing Abaqus Commands from External Model.** Performance enhancements have been made for importing data from External Model into Mechanical for the Abaqus *SURFACE and *NSET/*ELSET commands.

Abaqus *SURFACE Command

Previously, the presence of numerous *SURFACE commands in the input file was causing performance degradations. Refactoring enhancements that handle *SURFACE commands have significantly improved processing speeds, ranging from 35% to 95% improvements. Note the following:

- For 100-200 *SURFACE commands: 35% - 40% improvement.
- For 400-500 *SURFACE commands: 45% - 95% improvement.

Abaqus *NSET/*ELSET Commands Ending with a Comma

Previously, when either the *NSET or *ELSET commands ended with a comma, the application interpreted it as a missing entity and gathered unnecessary command information. This inefficient data collection process has been refactored and the enhancements have led to improvements of up to 98%.

4. Add-ons

The following updates were made to the Mechanical add-ons for Release 2024 R2:

- The **DesignLife** add-on now offers the following capabilities:
 - **Frequency Selection Method** for frequency-domain loading events.
 - Selection of existing **Harmonic** and **Modal** systems for the **Environment** definition of a frequency-based load.
 - Tabular display of **Shell Layer**, **Seam Weld Location** and **Seam Weld Type** results for a **Shell Seam Weld** analysis.

- **Temperature** option for **Strain** and **Stress** analyses.
- Additional **Surface Finish** options are now available for **Material Assignment**.
- Reading of the nCode **Material** database for **Stress**, **Strain**, **Gray Iron**, and **Safety Factor** analysis.
- **Alternating Stress**, **Mean Stress**, and **Minimum and Maximum Stress** can be returned under **Other Results** for all analysis types.
- The most-damaged node or mesh Ids can be extracted using the post graphics **Most Damaged Id** tabular information.
- The **Time Series Output History** now works in the **Time** and **Frequency** domains for multiple analysis types (**Stress**, **Strain**, **Gray Iron**, **Shell**, and **Solid Seam Weld**) and can be displayed for the selected **Most Damaged** Ids.
- **Fatigue Life** estimation for **MSUP Transient** analysis with manual **.rst** selection.
- The **NVH Toolkit** add-on now includes support for:
 - Non-unitary force values for modal and harmonic analysis in the **FRF Calculator**.
 - Cyclic symmetry using the **MAC Calculator**.
 - Coordinate transformation and differing units for RST to RST comparison in the **MAC Calculator**.
 - New DPF operators to improve performance of **MAC Table** calculation (both in **MAC Calculator** and **Pre-Test Calculator**).
 - The **FRF Calculator** is now available in a **Harmonic Analysis** for the **Mode Superposition** solution method.
 - The **FRF Calculator** now supports a load scaling factor in both **Modal** and **Harmonic Analysis**.
- **Restart Analysis** is a new addition to the Add-ons Ribbon. This add-on makes Static-Static or Transient-Transient structural restart capabilities available within the Mechanical environment.
- Additional localization support is available for the **Restart Analysis** and **Forced Response** add-ons. These add-ons now support Chinese, Japanese, German, and French languages as well as varied decimal separators.

5. General

The following general enhancements were made at Release 2024 R2:

- **License Preference Migration.** Now, when you update your version of the Mechanical application, your current Mechanical and Meshing license options are automatically migrated to the new version to make sure you maintain your desired license configuration.

- **Unit Systems.** A new unit system is now available: **Metric (mm, kg, kN, ms, mV, mA)**. All analyses that use this unit system and that have the **Solver Target** property setting, **Mechanical APDL**, the application sets **nmm** for the **Solver Unit System** property (**Analysis Settings > Analysis Data Management**).

6. Graphics

The following graphical enhancements were made at Release 2024 R2:

- **Accelerated Graphics/Animation.** In order to display more accurate lighting for results with extremely large deformations, the **Options** dialog, **Graphics** group, preference, **Use Faceted Lighting for Accelerated Graphics/Animation**, is now active (set to **Yes**) by default. The concession for this setting is that models, typically those with low resolution meshes, may produce a slightly more faceted display.
- **Linux Support.** The following graphical features are now supported on the Linux platform:
 - Accelerated Graphics Feature
 - Accelerated Animation Feature
 - Anti-Aliasing (MSAA) Preference
 - Display Graph Option
 - Probe Tool and Hit Point Coordinate Preference
- **Text Display Enhancements.** The quality of the text displayed in the **Geometry** window has been improved this release.

7. Geometry

The following geometry enhancements were made at Release 2024 R2:

- **Importing External Mesh Files.** Now, when you open Mechanical independently, you can import LS-DYNA Input (.k and .key), or Abaqus (.inp) files directly into the application using the **Add Model Import** option available from the **Geometry Imports** object.
- **Body Merge.** A new **Body Merge** object allows you to combine multiple geometric bodies with shared faces into a single merged body. The mesh is also merged if the model has been meshed. Body Merge simplifies complicated geometries by reducing the number of bodies in a model, which is useful when working with complex parts. It can also be used to merge bodies that were previously sliced to generate a hex dominant mesh.

8. External Model

The following enhancements for importing external model data were made for 2024 R2:

- **Worksheet Enhancements for Imported Mesh Data.** Several changes have been made to the worksheet that presents data for imported mesh data:

- The **Check/Uncheck** column of the worksheet has been renamed **Active** to represent the function of the column more clearly – activating and/or deactivating the data of a worksheet row.
- You can now select multiple worksheet rows and select the **Active** check box of any one of the rows to change the status of all of the rows.
- A new context (right-click) menu option, **Select Items**, enables you to automatically select all rows in the currently displayed page of the worksheet.

9. Contact and Connections

The following contact and connection enhancements were made at Release 2024 R2:

Contact Enhancements

- **Contact Split Program Controlled Behavior Change.** When the **Contact Split (DMP)** property (**Analysis Settings > Advanced**) is set to **Program Controlled**, the application now automatically performs contact pair splitting during the solution when the number of contact regions is less than the number of:
 - Cores when the **Distributed** option is enabled.Or...
 - Processes when the **Hybrid Parallel (Mechanical APDL)** option is **also** enabled.

In prior releases the application always performed contact split when **Contact Split (DMP)** property was set to **Program Controlled**.

- **Rigid-To-Flexible Contact.** The application no longer creates a duplicate layer of target elements (called an insulation layer) for contact regions defined between a rigid body surface and a flexible body. This optimization significantly improves solution processing performance for large assembly models.
- **Cylindrical Smoothing.** When you set the **Contact Geometry Correction** or **Target Geometry Correction** properties to **Smoothing** and the **Orientation** property to **Program Controlled**, you can now select multiple cylindrical faces for the scoping of the **Contact Region**.

Connections Enhancements

- **Connections Manager.** This new extension provides capabilities to automatically generate and manage connection objects for an entire model or a group of bodies within a model.

10. Mesh

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

11. Additive Manufacturing Process Simulation

See [Additive Manufacturing](#).

12. Acoustics Analyses

For Release 2024 R2, the following enhancements were made for [Acoustics Analyses](#):

- **Harmonic Acoustics Adaptive Mesh Display.** The [Morphing Region](#) object, available for [Harmonic Acoustics](#) analyses, now provides a new graphical display property: **Display Mesh**. You use this new option to display the adapted or "morphed" mesh of the **Morphing Region**, for a specified frequency value. You can change frequency values to review how the mesh adapts.

13. Coupled Field Analyses

For Release 2024 R2, the following enhancements were made for [Coupled Field Analyses](#):

- **Update View Factor.** Now, when you are performing a **Coupled Field Static** or a **Coupled Field Transient** analysis that specifies structural and thermal physics, the [Advanced](#) category of the **Analysis Settings** object includes a new property: **Update View Factor**. For each [Radiation](#) loading condition included in the analysis that has the **Correlation** property set to **Surface to Surface**, the application updates the view factors at the specified interval. The use of this property requires that you specify large deflection and at least one Radiation load.
- **Importing Loads from Maxwell.** You can now import **Surface Force Density** and **Body Force Density** loading conditions from the Maxwell application to downstream Coupled Field analyses.
- **Imported Body Temperatures.** You can now import temperatures from upstream **Coupled Field Static/Coupled Field Transient** analyses into downstream **Static Structural/Transient Structural** analyses.
- **Element Definition for Piezoelectric Coupling.** For Coupled Field analyses defined with piezoelectric coupling, the lower-order mesh now uses the element [SOLID225](#) instead of element [SOLID226](#) (high-order) with the mid-side nodes dropped.
- **Normal Stiffness in Coupled Field Analyses.** For electrostatic-structural **Coupled Field Static** and **Coupled Field Transient** analyses that include contact, you can now specify the **Program Controlled** setting for the **Normal Stiffness** property.

14. Analysis

For Release 2024 R2, the following analysis enhancements were made:

- **Including Objects in Multiple Analyses.** For environments that use the Mechanical APDL solver and that are independent (no initial conditions) from other environments, the **Analyses Selection** worksheet now supports **Material Assignment**, **Contact**, and **Joint** objects in addition to **Bolt Pretension** load objects. For all of these object types, you can specify whether to include or exclude the corresponding data for a simulation using the **Analyses Selection** worksheet.
- **Krylov Solution Method.** For [Harmonic Response](#), pure acoustic [Coupled Field Harmonic](#), and pure acoustic [Harmonic Acoustics](#) analyses, there is a new option for the **Solution Method** property: **Krylov**. The frequency-sweep harmonic analysis based on the Krylov method provides a high-performance solution for forced-frequency simulations in acoustic analyses. It can provide quick, estimated results compared to the **Full** method.

- **Iterative Solver Type Support for Harmonic Analyses.** Now, you can use the **Iterative** solver (**Analysis Settings > Solver Controls > Solver Type**) for **Coupled Field Harmonic** (pure acoustics), **Harmonic Response**, and **Harmonic Acoustics** (pure acoustics) analyses when the **Solution Method** property (**Analysis Settings > Options**) is set to **Program Controlled** or **Full**.
- **Welding Toolbox** is a native extension that allows you to manage welding process simulation. This tool eases setup in Mechanical and helps to optimize weld sequence and process parameters.
- **Rotordynamics** is a native extension that streamlines and accelerates the creation of acceptance documents for rotating machinery design. These capabilities can be used to insert pre and post processing objects as part of a vertical application for the rotordynamics industry.

15. Fracture

The following **Fracture** analysis enhancements were made at Release 2024 R2:

- **Crack Initiation and Propagation using SMART Crack Growth.** During a 3D Static Structural analysis, the **Fracture** analysis now provides a new object: **Crack Initiation**. You use this new object to specify a criterion for determining when the application initiates a crack in a specific region (or zone) of the geometry. You can also specify the location, orientation, and the size of the crack to be initiated. You apply this object and its definitions to a **SMART Crack Growth** object using the **Initial Crack** property in order to study the crack growth.
- **SMART Crack Growth.** For 3D Static Structural analyses, you can now scope a **Pressure** load to a **SMART Crack Growth** object. This enables you to apply the pressure to only the new crack faces the application generates during the smart crack growth process or to both the new and the initial crack faces of the crack associated with the **SMART Crack Growth** object.

16. Structural Optimization

The following **Structural Optimization** analysis enhancements were made at Release 2024 R2:

- **Refactoring Enhancements.**
 - For the **Mixable Density** optimization method, Ansys has refactored the material interpolation when doing Modal analyses in order to not create spurious modes due to intermediate densities.
 - For the **Mixable Density** and **Level Set** based optimization methods, the calculation of **Reaction Force response constraint**, with respect to optimized regions of the model, has been refactored to improve the accuracy.
 - For the **Mixable Density**, **Density Based**, and **Level Set** based optimization methods, the accuracy of stress results, specifically Maximal Principal Stress, has been improved. As a result, you may notice changes to displayed result values compared to previous releases.
- **Manufacturing Constraint.** A new **Manufacturing Constraint** is available **Complexity Index**. This manufacturing constraint enables you to control the complexity of optimized designs and minimize the creation of overly complex structures.

- **Primary Criterion for Modal Analyses.** Now, when you specify a **Primary Criterion** object under the **Solution** object of a Modal analysis, the **Frequency Reduction** property includes a new option: **Mode Tracking**. This option enables you to track an eigenmode of interest.
- **Mixable Density.** This optimization method now supports:
 - Upstream **Harmonic Response** analyses.
 - **User Defined Criteria** in upstream **Harmonic Response** analyses that you can use as a **Response Constraint** and **Objective**.
 - The **Stress** criterion that you can use as a **Response Constraint** or **Objective**.
- **Shape Optimization.** This optimization method now supports the:
 - Use of the **Extrusion** Manufacturing Constraint.
 - **Local Equivalent von-Mises Stress** Response Constraint.

17. Motion

The following enhancements were made to Motion at Release 2024 R2:

- **Velocity** and **Acceleration** **Waterfall** diagrams are now available for Motion analyses, using a Short Time Fourier Transform (STFT) algorithm.
- Motion now allows the use of **Condensed Part** and **Imported Condensed Part** objects, including support for contact.
- **Data View** is now available for Spline objects, allowing rapid transfer of data via copy and paste.
- **Considering Contact Area** is now available for contact Stiffness Type, where the contact stiffness is proportional to the contact area.
- Additional **Mass Information** parameters have been added to the **Drivetrain Toolkit** to allow more accurate calculation of Gear Web mass and moment of inertia.

18. Loads/Supports/Conditions

The following loads/supports/conditions enhancements were made at Release 2024 R2:

- **Fluid Penetration Pressure.** For Static Structural analyses, there is a new loading condition available: **Fluid Penetration Pressure**. This loading condition enables you to simulate surrounding fluid or air penetrating into a contact interface.
- **Pressure.** For 3D Static Structural analyses, you can now scope a **Pressure** load to a **SMART Crack Growth** object. This enables you to apply the pressure to the new crack faces the application generates during the smart crack growth process or to these new crack faces in addition to the initial crack faces of the crack associated with the **SMART Crack Growth** object.
- **Specifying Loads with Tables.** This feature has the following changes:

- You can now directly select table names when you are specifying magnitudes for [Pressure](#), [Temperature](#), and [Thermal Condition](#) boundary conditions.
- The **Add Table** option on the Magnitude menu automatically creates a table with an appropriate dependent variable and the independent variable time.

19. Imported Loading Conditions

The following imported loading condition enhancements were made at Release 2024 R2:

- **Importing Load Data from External Data.** You can now use the **External Data** feature to import velocities from Hierarchical Data Format (HDF) Version 5 (.hdf5) files that contain the results and meshed regions from external solvers, such as MASTA, NATRAN, etc., using [Imported Velocity](#). This feature is supported for [Coupled Field Harmonic](#) and [Harmonic Acoustics](#) analyses only.
- **Imported Remote Loads from Maxwell.** Now, when you import remote loads from a Maxwell analysis:
 - And the analysis includes a parametric sweep over RPMs, the [Imported Remote Loads](#) object includes two new properties:
 - **Interpolate RPM:** This property enables you to linearly interpolate the harmonic force and moment loads between two known RPM values with equal spacing. You can also specify custom spacing using a manually defined comma-separated values (.csv) file.
 - **Filter by Order:** This property enables you to import only specific orders for each RPM.
 - **Importing Loads from Maxwell.** You can now import **Surface Force Density** and **Body Force Density** loading conditions from the Maxwell application to downstream Coupled Field analyses.
- **Imported Remote Loads.** For **Imported Remote Loads** feature, when you generate globally available remote points and select the option to **Reuse Remote Points**, the application now includes all global remote points to avoid duplication.
- **Imported Body Temperatures.** You can now import temperatures from upstream **Coupled Field Static/Coupled Field Transient** analyses into downstream **Static Structural/Transient Structural** analyses.
- **Importing Loads from Ansys Rocky.** Now, when you import data from Ansys Rocky into Mechanical, the data for **Imported Convection** and **Imported Force** loads automatically populates the **Data View** worksheet for the given load object.
- **Data View Worksheet Additions for Submodeling and Results File Import.** The **Data View** worksheet for [Imported Temperatures](#) and [Imported Body Temperatures](#) now supports the data properties **Scale** and **Offset** when importing boundary condition loads through the [Submodeling](#) and [Results File Import](#) features. These options enable you to modify imported temperature data by either multiplying the scale and/or adding an offset to the temperature values.

20. Solution

The following solution enhancements were made at Release 2024 R2:

- **Resource Prediction.** This feature is now only supported with the **Structures AI+** license.

Additionally, the feature now:

- Is supported for **Full** Harmonic Response analyses when the **Solution Method** property is set to either **Full** or **Program Controlled**.
- Supports predictions for Static Structural analyses that include nonlinear conditions such as large strain or deformation, nonlinear material property definitions, as well as nonlinearities associated with contact conditions.
- Performs predictions using the number of cores specified in the [Solve Process Settings](#) dialog. Previously, the application performed prediction and displayed table data based on a default core count of 4 cores.

21. Explicit Dynamics

The Workbench application provides two analysis systems that enable explicit solutions:

- The *Explicit Dynamics* system is a Mechanical Application-integrated provision of the Autodyn FE (Lagrange) and multiple-material Euler solvers plus Euler-Lagrange Coupling (providing FSI). The Autodyn solver is equivalent to the Explicit Dynamics Solver.
- The *LS-DYNA* and *LS-DYNA Restart* systems are Mechanical Application-integrated provisions of the LS-DYNA solver with extensive support through native Mechanical Application objects or keyword management.

The enhancements for these systems for Release 2024 R2 are described below.

Explicit Dynamics

For the Explicit Dynamics system, note the following enhancements for Release 2024 R2:

- **Intel MPI Upgrade.** On Windows and Linux, Intel MPI support has been upgraded to version 2021.11.

LS-DYNA

For LS-DYNA systems, note the following enhancements for Release 2024 R2:

- **Active Unit System.** LS-DYNA can now use the active unit system of Mechanical as the solving unit system. If the **Solver Units** property of the Solver Controls in the Analysis Settings is set to Active System, the solver unit system used for the input file generation will be the closest consistent unit system to the active unit system.
- **Split Input File.** LS-DYNA enables the input file to be split when the **Split Input File** option of the analysis Settings is activated. It will split the `input.k` file into 5 additional files (`control.k`,

database.k, mesh.k, contact.k, component.k), allowing the input file to be read more easily.

- **Coupled Structural Thermal Electromagnetic Analysis.** The LS-DYNA system in Mechanical lets you perform an analysis using electromagnetic Voltage and Current boundary conditions to evaluate the resistive heating effects in the model. Electromagnetic material properties can be added to materials.
- **ALE Enhancements.** Fluid structure interaction data is now available to the [Binout Tracker](#). By default, the LS-DYNA system now ensures ALE elements are evenly distributed to all processors.
- **LS-DYNA Acoustics Analysis.** LS-DYNA now provides for a Multi Load Steps Acoustic simulation using the BEM technology, and allowing coupling with Harmonic Response.
- **Additional Explicit Materials.** [Explicit materials](#) such as Air (Atmospheric) and most explosive materials are now available for the LS-DYNA system in Mechanical.
- **Additional Material Properties.** Ideal Gas EOS and Explosive JWL [material properties](#) are now available for the LS-DYNA system in Mechanical.
-

22. Results

The following result enhancements were made at Release 2024 R2:

- **Panel Contribution.** The **Panel Contribution** option now supports most [Frequency Response](#) results for **Harmonic Response** analyses.
- **Energy Contribution.** For **Modal** analyses only, you can now specify an **Energy Contribution** result for potential energies. This result produces a mosaic chart highlighting the potential energy for each body and every mode shape.
- **Waterfall Diagram Results.** Several changes have taken place for waterfall diagram results:
 - **Filtering by RPM and Order.** Now, using the options of the [Chart and Tabular Data](#) category of the Details pane, you can simultaneously filter waterfall diagram results by RPM **and** Order so you can further refine the result data presented in the **Graph** and **Tabular Data** windows. In the previous release, you could only filter these results by RPM **or** Order.
 - The **Modal Coefficient Factors (MCF) Waterfall Diagram** result has been renamed to **Mode Contribution**. You access this new result type from the new **Contribution** menu on the **Solution** tab for the supported analysis types.
 - The Details pane properties for waterfall diagram results are now available under a new category: **Waterfall Data**. Previously, these properties were included in the [Chart and Tabular Data](#) category. The behavior of the properties remains the same.
 - The [Chart and Tabular Data](#) category includes new filter properties (**Display Panel/Display Order**) based on analysis and result types. You can now display a mosaic chart for different contribution types. This enables you to perform targeted post processing by filtering complex response data to determine which bodies have the highest response at specific mode shapes and frequencies.

- The **Selection Mode** property of the [Chart and Tabular Data](#) category has been renamed to **Filter By**.
- **Krylov Residual Norm**. Now, when you solve a **Harmonic Response**, pure acoustic **Coupled Field Harmonic**, and pure acoustic **Harmonic Acoustics** analysis using the **Krylov** option of the **Solution Method** property, a new **Frequency Response** result is available: **Krylov Residual Norm**. This result plots error estimations (residuals) across a frequency range for the harmonic distribution. The plot can provide information on how to best split the whole frequency range into multiple ranges.
- **Analysis Settings > Output Controls**. The **Output Controls** category includes a new property: **Output Selection**. This property enables you to store results only on specific geometry- or meshed-based Named Selections in order to reduce result file size.
- **Reaction Probes**. You can now scope [Force Reaction](#), [Moment Reaction](#), and [Heat Reaction](#) probes to **Named Selections**.
- **Composite Failure Tool**. The **Composite Failure Tool** includes a new property: **Show On Reference Surface**. This property enables you to display the worst failure, per solid stack, on the shell mesh of the specified Reference Surface only.
- **Python Result**. When you insert a **Python Result**, the worksheet automatically includes default script content. In previous releases, this script content was based on the Total Deformation result. Now, the content depends on the type of analysis to which you have added the result.

Mechanical APDL Release Notes

Release 2024 R2 of the Mechanical APDL application offers most of the capabilities from prior releases plus many new features and enhancements. Areas where you will find updates and new capabilities include the following:

- [Structural \(p. 55\)](#)
- [Multiphysics \(p. 61\)](#)
- [Solvers \(p. 63\)](#)
- [Commands \(p. 65\)](#)
- [Elements \(p. 67\)](#)
- [Documentation \(p. 69\)](#)
- [Technology Showcase: Example Problems \(p. 69\)](#)

For more information about this release, see the [Update Guide \(p. 70\)](#) section of this document and the [Global Release Notes](#).

1. Structural

Release 2024 R2 includes new features and enhancements for the following structural analysis disciplines:

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

1.1. Contact

Release 2024 R2 includes the following enhancements for analyses involving contact:

- [1.1.1. Enhancements to Contact Friction Models](#)
- [1.1.2. Enhancements to Multipoint Constraints and Assemblies](#)
- [1.1.3. DOF-based Rigid Surface Constraints and Rigid Elements](#)
- [1.1.4. Performance Improvements to CNCHECK Command](#)
- [1.1.5. Performance Improvements for Fluid Penetration Pressure Loads](#)
- [1.1.6. Performance Enhancements related to Contact Surface Wear](#)

1.1.1. Enhancements to Contact Friction Models

Prior to this release, only the variational range of the normal contact stiffness can be controlled. In this release, the variational range of the tangential contact stiffness can be controlled as well. The default method of updating the tangential contact stiffness (`KEYOPT(13) = 0`) is suitable for most applications. However, the new option improves nonlinear convergence in complex frictional contact models.

1.1.2. Enhancements to Multipoint Constraints and Assemblies

Selecting a constraint type for assemblies is problem dependent and can have a significant impact on the solution. In this release, the automatic constraint type detection logic (`KEYOPT(5) = 0` of target element) has been revised so that larger percentages of assembly models can be solved by default with improved robustness and accuracy. For more information, see "[TARGE170 Input Summary](#)" in the [Mechanical APDL Element Reference](#).

1.1.3. DOF-based Rigid Surface Constraints and Rigid Elements

Defining a Pre-meshed Body as a Rigid Body with **RBGEN**

In most applications, rigid bodies are modelled as discretized finite elements in a meshed body. The rigid body can be defined on the exterior of a pre-meshed body discretized by solid, shell, and beam elements (called underlying elements).

From this release onward, a new **RBGEN** command automatically generates rigid target elements exterior to given element components along with a pilot node (**TSHAP,PILO**), and adds rigid nodes defined as pin-type or tie-type nodes using nodal components using **TSHAP,POINT**.

This command enables you to define a rigid body with one Mechanical APDL **RBGEN** command. It is helpful when converting a flexible body to a rigid body. It also reduces the input file size where rigid target elements are written as longlist using **EBLOCK**.

Tie and Pin node

This release introduces tie-type node and pin-type nodes to rigid body definition. Tie nodes include both translational and rotational degrees of freedom. Pin nodes only include translational degrees of freedom. Define a node segment as a pin node with `KEYOPT(7) = 1` in the target element. Define a node segment as a tie node with `KEYOPT(7) = 0` in the target element.

1.1.4. Performance Improvements to **CNCHECK** Command

Global Pinball for Contact Trim

When working with large assemblies and an excessive number of bonded or small sliding or symmetric contact/target elements, trim is a useful tool to reduce the model size. This release introduces "global pinball" in the **CNCHECK** command. This tool enables you to quickly apply the same pinball value to all the selected contact pairs during the trim process.

Auto-trim in Contact Split

Starting from this release, the **CNCHECK,DMP** command performs additional contact trim before the contact split operation. This is aimed to reduce contact split operation time and further reduce model size in solution runs.

1.1.5. Performance Improvements for Fluid Penetration Pressure Loads

Fluid pressure penetration loads are used to model when surrounding fluid or air penetrates into the contact interface from one or multiple locations based on contact status. The fluid penetration path detection algorithm has been improved to significantly reduce the computational time for three dimensional problems with fluid penetration pressure loads.

1.1.6. Performance Enhancements related to Contact Surface Wear

The performance of models with contact surface wear (**TB,WEAR**), where wear is modelled for post-processing purposes only and does not affect the solution (by defining **TBDATA,C5=- 99**), is improved when using contact surface splitting (**CNCHECK,DMP**).

1.2. Elements and Nonlinear Technology

Release 2024 R2 includes the following enhancements to elements and nonlinear technology used in structural analyses:

- 1.2.1. New Genb Joint Element
- 1.2.2. Mesh-Coarsening Enhancements for Nonlinear Adaptivity
- 1.2.3. Element Removal for Nonlinear Adaptivity
- 1.2.4. Direct Element Embedding Enhancement
- 1.2.5. Automatic Mesh Optimization of Base Mesh
- 1.2.6. Preprocessing Stress Evaluation of Shell Elements
- 1.2.7. CPT Element Support for Reinforcing
- 1.2.8. Improved Output Control for Selected Result Data
- 1.2.9. Gasket Element Enhancements

1.2.1. New Genb Joint Element

A new joint element is available with relative rotations defined by the 3-1-3 Euler angles. Use the new **MPC184-Genb** joint element to connect two bodies to transmit motion from one body to another. Stiffness and damping behavior, locks and stops, joint loads and boundary conditions can be specified for this joint as needed.

1.2.2. Mesh-Coarsening Enhancements for Nonlinear Adaptivity

You can now use the energy-based and location-based criteria with mesh-coarsening in a nonlinear adaptivity analysis. The added capability enables you to apply mesh-coarsening on unnecessarily fine meshed regions or on regions with no critical features. Coarsening remeshings of debonding problems (cohesive-zone material) using the contact-based criteria have been improved, especially

for subsequent remeshings. For more information, see [Nonlinear Mesh Adaptivity in the *Nonlinear Adaptivity Analysis Guide*](#).

1.2.3. Element Removal for Nonlinear Adaptivity

You can now use the new element-removal-based criterion (**NLADAPTIVE,,,RELM**) to remove elements from a deformed body in a nonlinear adaptivity analysis. The added capability determines which element(s) to remove based on the maximum equivalent strain measure or maximum principal strain measure of the element(s). This feature also allows you to manually define the candidate elements for removal. You can use the new **NLMESH,DCON** control to generate contact and target elements in gaps created in the element removal process. For more information, see [Nonlinear Mesh Adaptivity Criteria in the *Nonlinear Adaptivity Analysis Guide*](#).

1.2.4. Direct Element Embedding Enhancement

The **LINK228** 3D Coupled-Field Link element with **KEYOPT(1) = 101** and **1001** is now supported for the direct-embedding workflow. For more information, see [Direct-Embedding Workflow in the *Structural Analysis Guide*](#).

1.2.5. Automatic Mesh Optimization of Base Mesh

A new **MSHOPTIM** command is available for automatic mesh optimization. This action command tries to generate an optimum base mesh in the selected mesh region based on the cross-sectional sizes of intersecting embedded elements. The optimized base mesh can be used in subsequent [reinforcing or direct element embedding](#) modeling approaches to overcome simulation accuracy issues.

1.2.6. Preprocessing Stress Evaluation of Shell Elements

Perform preprocessing-level stress evaluation of structural 3D shell elements using the new **EENS** command. This action command calculates element results (such as strains and stresses) based on the curvature that is caused by bending or twisting. Membrane deformation is ignored. This command can be used to evaluate element solutions without running a regular finite element simulation, which may be useful during the design phase of a product that requires frequent shape changes.

1.2.7. CPT Element Support for Reinforcing

You can now use coupled pore-pressure-thermal mechanical solid elements (**CPTnnn**) as base elements in a reinforcing analysis using the [mesh-independent method](#).

1.2.8. Improved Output Control for Selected Result Data

The **OSRESULT** command for controlling the output quantities written to the database has been enhanced. New Item and Component labels have been added for more robust result-data control.

1.2.9. Gasket Element Enhancements

The **INTER194** and **INTER195** 3D gasket elements with thin solid option (**KEYOPT(2) = 2**: through-the-thickness, in-plane membrane, and transverse-shear deformation) now support von Mises

plasticity models with [isotropic hardening](#) or [kinematic hardening](#) (**TB**,CHABOCHE; **TB**,NLISO or **TB**,PLASTIC).

1.3. Material and Fracture Modeling

Release 2024 R2 includes the following enhancements to material modeling and fracture analysis technology used in structural analyses:

- [1.3.1. Python Extension for Ansys Materials Lab](#)
- [1.3.2. Material Curve-Fitting Enhancements](#)
- [1.3.3. Initial Deformation for Hyperelastic Materials](#)
- [1.3.4. Cyclic-Loading Enhancements](#)
- [1.3.5. Inline Material Criterion for Adaptive Crack Initiation](#)
- [1.3.6. von Mises Plasticity Enhancement](#)
- [1.3.7. Support for Rotated Nodal Coordinate Systems in SMART Remeshing Zone](#)
- [1.3.8. Support for Node and Element Components Tracking and Updating During SMART Remeshing](#)

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. Python Extension for Ansys Materials Lab

AML Python Module is the Python extension for Ansys Materials Lab (AML) that allows you to access Ansys structural materials to:

- Evaluate material tangent and stress for supported material models.
- Perform parameter fitting for supported material models.

You can use AML Python Module for parameter studies, yield surface generation, and on-the-fly stress generation. You can also develop your own parameter fitting tools by creating custom experiments and using SciPy-like tools for optimization.

For more information, see [Using AML Python Module in the Material Reference](#).

1.3.2. Material Curve-Fitting Enhancements

All material curve-fitting now occurs via the Ansys Materials Lab (AML) framework and adheres to the same command structure.

Support is now available for anisotropic hyperelastic material models. Anisotropic hyperelasticity allows for uniaxial, biaxial, and volumetric loading conditions with load-direction information.

Creep models (with elasticity) now support both creep strain data-fitting and uniaxial loading.

AI-based parameter initialization is now available for Prony series models with shear modulus and/or bulk modulus data in both time domain and frequency domain using the nearest-neighbor method. An optional feed-forward neural network method for obtaining a starting point for the nonlinear regression process is also available.

For more information, see [Material Curve-Fitting in the *Material Reference*](#).

1.3.3. Initial Deformation for Hyperelastic Materials

Due to the nonlinear behavior of hyperelasticity-based finite-strain material models, initial deformation cannot be directly defined consistently via initial stresses. Now, however, you can apply an initial-deformation gradient to indirectly define an initial-stress state for those material models. A typical application for the initial-deformation gradient involves an acoustic or fluid-structure interaction (FSI) analysis where the correct stiffness of a hyperelastic material in an updated deformed configuration is required. For more information, see [Applying Initial Deformation Gradient in the *Advanced Analysis Guide*](#).

1.3.4. Cyclic-Loading Enhancements

When specifying time-stepping parameters in a [cyclic-loading analysis](#), you can now modify default time-stepping and predictor behavior when appropriate for a given analysis (for example, when many closely-spaced defined time points are needed to describe the loading and a smooth transition occurs from one defined point to the next). For more information, see [Step 5. Specify Time-Stepping Parameters in the *Advanced Analysis Guide*](#).

Cyclic-loading and cycle-jump analyses now support [multiframe restart](#) without the need to redefine and reapply cyclic-loading tables. A multiframe restart can resume a job at any point in a cyclic-loading or cycle-jump analysis for which information is saved. For more information, see [Restarting a Cyclic-Loading Analysis](#) and [Restarting a Cycle-Jump Analysis in the *Advanced Analysis Guide*](#).

1.3.5. Inline Material Criterion for Adaptive Crack Initiation

A new **ADPCI** command option (*Action* = FCRI) enables you to define an inline material criterion for initializing a crack. The new capability overrides the crack-initiation criterion defined via the material data table (**TB**,CRKI) and **ADPCI**,DEFINE,,,MATID.

1.3.6. von Mises Plasticity Enhancement

You can now use von Mises plasticity models with [isotropic hardening](#) or [kinematic hardening](#) (**TB**,CHABOCHE; **TB**,NLISO or **TB**,PLASTIC) with [INTER194](#) and [INTER195](#) 3D gasket elements with thin-solid option (KEYOPT(2) = 2: through-the-thickness, in-plane membrane, and transverse-shear deformation).

1.3.7. Support for Rotated Nodal Coordinate Systems in SMART Remeshing Zone

The SMART crack-growth method now supports [displacement boundary conditions](#) applied to nodes with their nodal coordinate systems rotated (specified via **NROTAT**, **NMODIF**, **NANG**, or **NORA**). The node rotations and boundary condition values are properly mapped during remeshing.

Note that any rotations of nodal coordinate systems are maintained inside the remeshing zone only if those nodes are associated with a displacement boundary condition. Otherwise, the nodal rotations are not maintained during remeshing, and any new node is set with the default nodal coordinate system.

1.3.8. Support for Node and Element Components Tracking and Updating During SMART Remeshing

The SMART crack-growth method now tracks and updates user-defined components when the component update key is activated with the **CM,,,,,KOPT=1** command, but only for solid element and surface node components. This feature enables you to adjust or revise the loading parameters as needed over an element or node component after remeshing. For additional information and limitations, see [SMART Crack-Growth Assumptions and Limitations in the *Fracture Analysis Guide*](#).

1.4. Linear Dynamics

Release 2024 R2 includes the following enhancements for structural analyses involving linear dynamics:

1.4.1. Harmonic Balance Method (HBM) Cyclic Analysis

1.4.2. Fixed-interface CMS Method for Single Stage Sector Models

1.4.1. Harmonic Balance Method (HBM) Cyclic Analysis

The harmonic balance method (HBM) cyclic analysis enables efficient calculation of multiharmonic forced responses for cyclically symmetric systems with local nonlinearities. This technology calculates the steady-state responses of cyclically symmetric structures to engine order (traveling wave) type excitation. A traveling wave excitation can be applied as nodal forces or as surface pressure loads. **Macros** are available to facilitate modeling of the cyclically symmetric structure using single stage (**MSOPT**) cyclic superelements with different spatial harmonics generated from single sector models. Use an HBM cyclic analysis to simulate the effects of nonlinearities (including contact interfaces) as well as prestress (such as rotational velocity, temperature, and contact pressure) in rotating and turbo machinery. See [HBM Cyclic Procedure in the *Harmonic Balance Method Analysis Guide*](#) for more details and examples.

1.4.2. Fixed-interface CMS Method for Single Stage Sector Models

The fixed-interface component mode synthesis (CMS) method can now be applied to single stage (MSOPT) sector models. Prestress effects can be included using the linear perturbation substructuring procedure. CMS for single stage is an efficient way to reduce the size of a cyclically symmetric model and thus significantly reduce the solution time (use pass) in advanced workflows. See the [Substructuring Analysis Guide](#) and the [Multistage Cyclic Symmetry Analysis Guide](#) for details about the procedure.

2. Multiphysics

Release 2024 R2 includes the following enhancements for analyses involving multiphysics environments:

2.1. Acoustics

2.2. Thermal

2.3. Coupled-Field

2.1. Acoustics

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

2.1.1. Harmonic Nonlinear Acoustics Solver Model

2.1.2. Difference Frequency Method

2.1.1. Harmonic Nonlinear Acoustics Solver Model

For nonlinear acoustics, a harmonic solver model based on the pressure-based Westervelt equation is now available for 2D, axisymmetric, and 3D acoustics elements, including [FLUID243](#), [FLUID244](#), [FLUID30](#), [FLUID220](#), and [FLUID221](#). For details, see [Nonlinear Acoustics Governed by the Westervelt Equation for Full Harmonic Analysis](#) in the Acoustic Analysis Guide and [The Finite Element Model for Harmonic Analysis](#) in the Theory Reference.

2.1.2. Difference Frequency Method

For the acoustic signal at the difference frequency produced by the nonlinear interaction of two primary ultrasound waves, a harmonic solver model based on the pressure-based Westervelt equation is now available for 2D, axisymmetric, and 3D acoustics elements, including [FLUID243](#), [FLUID244](#), [FLUID30](#), [FLUID220](#), and [FLUID221](#). For details, see [Difference-Frequency Generation in Nonlinear Acoustic Waves Governed by the Westervelt Equation](#) in the Acoustic Analysis Guide and [Difference-Frequency Generation in Nonlinear Acoustic Waves](#) in the Theory Reference.

2.2. Thermal

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

2.2.1. 3D Thermal Layered Shell Element: SHELL294

2.2.1. 3D Thermal Layered Shell Element: SHELL294

A new thermal layered shell element is available for 3D steady-state or transient thermal analyses. [SHELL294](#) has in-plane and through-thickness thermal conduction capabilities. It generates temperatures that can be passed to structural shell elements to model thermal bending. The element has four nodes with no limitation on the number of interpolation layers (defined by through-thickness DOFs) whereas the corresponding legacy element, [SHELL131](#), is limited to 31 layers. In addition, the new element is geometrically more accurate than its legacy counterpart. It distinguishes between the bottom and top faces in terms of their area and spatial location when applying surface loads, computing view factors, and handling contact. It also considers geometry when binding adjacent elements with different layering, such as at a joint or at the runout of a tapered layer. Furthermore, it can have several material layers per interpolation layer for a better balance between accuracy and computational load. Lastly, it is more robust as it can be combined with other element types with different sets of DOFs in the same model. For verification examples, see [VM97](#) and [VM271](#) in the *Ansys Mechanical APDL Verification Manual*.

2.3. Coupled-Field

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

2.3.1. Enhanced Strain Formulation Supported for PLANE222 and SOLID225 for Structural-Electric-Diffusion Analysis

2.3.2. Output Flux Density Vectors for Coupled-Diffusion Analyses

2.3.3. Circuit Modeling Enhancement

2.3.1. Enhanced Strain Formulation Supported for PLANE222 and SOLID225 for Structural-Electric-Diffusion Analysis

You can now enable the enhanced strain or the simplified enhanced formulation in a structural-electric-diffusion analysis (KEYOPT(1) = 100101) by setting KEYOPT(6) = 2 or 3, respectively, on [PLANE222](#) and [SOLID225](#) coupled-field elements. The enhanced strain formulation improves solution accuracy in bending-dominated and nearly incompressible problems by preventing shear and volumetric locking, respectively.

2.3.2. Output Flux Density Vectors for Coupled-Diffusion Analyses

Coupled-diffusion analyses using [PLANE222](#), [PLANE223](#), [SOLID225](#), [SOLID226](#), and [SOLID227](#) now output flux density vectors corresponding to pure diffusion, stress migration, thermomigration, and electromigration. The components and magnitude of the flux density vectors are output as SMISC items 2 through 17. The new output quantities can be used to determine the contribution of each individual migration mechanism (concentration gradient, hydrostatic stress gradient, thermal gradient, electric field) to the total diffusion flux.

2.3.3. Circuit Modeling Enhancement

The resistance (R), capacitance (C), and inductance (L) real constants with [CIRCU94](#) and [CIRCU124](#) elements can now be defined using tabular input as functions of time or frequency in a transient or harmonic analysis, respectively. The ability to update time- or frequency-varying circuit parameters within the solution improves efficiency.

3. Solvers

Release 2024 R2 includes the following improvements to the solution process:

- [3.1. Distributed-Memory Parallel Processing Enhancements](#)
- [3.2. Enhanced Memory Reduction for Models with Constraint Equations and High Core Counts](#)
- [3.3. Reduced Peak Memory Usage for Uncompressed Node and Element Numbering](#)
- [3.4. Support for Devices for the GPU Accelerator](#)
- [3.5. Improved Performance for the PCG Solver](#)
- [3.6. Support for Mixed u-P Formulation with the PCG Solver](#)
- [3.7. Beta Feature - Frequency Domain Decomposition for Modal Analyses](#)

3.1. Distributed-Memory Parallel Processing Enhancements

Enhancements include:

- Modifications to the default processor pinning strategy and communication protocols (supplied to Open MPI as command-line options) yielded competitive performance on both single machines and HPC clusters with AMD-based processors.
- The default support for message passing interface (MPI) software has been upgraded to Intel MPI 2021.11.0 for both Linux and Windows. In prior releases, the default was Intel MPI 2021.10.0.

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

3.2. Enhanced Memory Reduction for Models with Constraint Equations and High Core Counts

The modified architecture of constraint equations significantly reduces memory for models containing a large number of constraint equations and using high core counts (32 or more cores).

3.3. Reduced Peak Memory Usage for Uncompressed Node and Element Numbering

The internal data structures used to store number mappings have been redesigned, with a particular focus on both node and element number mappings. This resulted in a significant reduction of Mechanical APDL peak memory usage when the numbering is highly uncompressed, for example when the maximum defined node or element number is many times greater than the number of defined nodes/elements.

3.4. Support for Devices for the GPU Accelerator

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

- NVIDIA L40
- NVIDIA RTX 6000 Ada
- NVIDIA RTX 5000 Ada
- NVIDIA RTX A5500
- NVIDIA RTX A4500

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

3.5. Improved Performance for the PCG Solver

The performance of the PCG solver when using one or more GPUs has been improved. Starting in this release, unsymmetric matrices are supported with the PCG solver when using one or more GPUs. Also, when using the PCG Lanczos eigensolver, additional calculations have been offloaded to the GPU for acceleration, which further improves the performance.

3.6. Support for Mixed u-P Formulation with the PCG Solver

For static analyses and transient analyses that use the full method, the PCG solver now supports the mixed u-P element formulation option for the following element types: [PLANE182](#), [PLANE183](#), [MPC184](#), [SOLID185](#), [SOLID186](#), and [SOLID187](#). This is the new default for the *LM_KEY* option of **PCGOPT** (*LM_KEY* = OFF).

3.7. Beta Feature - Frequency Domain Decomposition for Modal Analyses

Modal analyses now support the use of frequency domain decomposition (**DDOPTION,FREQ**) as a beta feature. For details, see [Frequency Domain Decomposition for Modal Analyses in the Mechanical APDL Beta Features](#).

4. Other Enhancements

The following other enhancements have been added:

[4.1. Enhancement for Element-Single Value Results](#)

[4.2. CAD Support Updates](#)

4.1. Enhancement for Element-Single Value Results

Element-single value results can now be used with [INTER194](#) and [INTER195](#) 3D gasket elements with thin solid option KEYOPT(2) = 2. For more information, see [Element Single-Value Results](#).

4.2. CAD Support Updates

The following CAD support updates are available in Mechanical APDL.

- NX 2312 support for:
 - Red Hat Enterprise Linux 8
 - SuSE Linux Enterprise Server and Desktop 15
- ACIS 2024 (Windows)
- CATIA V5-6R2024 (Windows)

For more information on CAD support in Mechanical APDL, see [Introduction to the Connection Functionality in the Connection User's Guide](#).

5. Commands

This section describes changes to commands at Release 2024 R2:

[5.1. New Commands](#)

[5.2. Modified Commands](#)

[5.3. Undocumented Commands](#)

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

5.1. New Commands

The following new commands are available:

- **EENS** – Evaluates element solutions based on assumed element deformation.
- **MSHOPTIM** – Optimizes a selected mesh based on predefined finite element specifications.
- **RBGEN** – Creates rigid bodies from pre-meshed element components.
- **SCCONNECT** – Specifies system coupling connection parameters.

5.2. Modified Commands

The following commands have been enhanced or otherwise modified:

- **ADPCI** – Defines parameters associated with adaptive crack initiation. A new option (*Action* = FCRI) enables you to define an inline material criterion for initializing a crack (instead of creating a new material data table via **TB,CRKI**).
- **AWAVE** – Specifies input data for an acoustic incident wave. Use the new option, *KCN*, to designate a coordinate system for the direction of normal velocity for a back enclosed loud speaker (*Wavetype* = BACK).
- **EBLOCK** – Defines a block of elements. A new **COMPACT** keyword makes the elements part of a compact format where the first field is the element number and the remaining fields are the node numbers.
- **CDWRITE** – Writes geometry and load database items to a file. A new option (**CDWRITE**,,,,,,BINARY) is available to write .cdb files in binary format when applicable. The blocked format is used in the other cases.
- **CINT** – Defines parameters associated with fracture-parameter calculations. A new **SMOOTH** option is available for fracture-parameter data smoothing.
- **CLOAD** – Initiates a [cyclic-loading analysis](#). The command now enables you to modify default time-stepping and predictor behaviors when appropriate for a given analysis.
- **CMACEL** – Specifies the translational acceleration of an element component. A new option (**CMACEL**,,DELETE) is available to delete the translational acceleration from all components.
- **CMOMEGA** – Specifies the rotational velocity of an element component. A new option (**CMOMEGA**,,DELETE) is available to delete the rotational velocity from all components.
- **CMDOMEGA** – Specifies the rotational acceleration of an element component. A new option (**CMDOMEGA**,,DELETE) is available to delete the rotational acceleration from all components.
- **CNCHECK** – Quickly apply the same pinball value to all the selected contact pairs during the trim process, and perform additional contact trim before the contact split operation.
- **/CONFIG** – Assigns values to Mechanical APDL configuration parameters. A new option (**/CONFIG**,RE-SUPREC) is available to control the precision when the program writes the results file.
- ***DIM** – Defines an array parameter and its dimensions. The argument *PAR* now supports up to 240 characters for table parameter names. The Mechanical APDL GUI cannot fully display parameter names that are longer than 32 characters.

- **HBMOPT** – Specifies solution options for a [harmonic balance method \(HBM\)](#) analysis. A new option (**HBMOPT,CYCLIC**) is available to activate an HBM cyclic symmetry analysis.
- **HROPT** – Specifies harmonic analysis options. The command now supports a new harmonic non-linear acoustic method, *Method* = HNLA.
- **INISTATE** – Defines initial-state data and parameters. The command can now apply an [initial-deformation gradient](#) for hyperelasticity-based material models.
- **MP** – Defines a linear material property as a constant or a function of temperature. Two new material property labels (BETW and SDIF) are available to define terms used in the Westervelt Equation for harmonic nonlinear acoustic analyses.
- **OSRESULT** – Controls the selected result data written to the database. New Item and Component labels have been added for more robust result-data control.
- **PCGOPT** – Controls PCG solver options. The PCG solver now supports certain PLANE/SOLID/MPC element types that use the mixed u-P (Lagrange multipliers) formulation when *LM_Key* = OFF (new default).
- **RSURF** – Generates the radiosity surface elements and stores them in the database. Use the new *DKEY* argument to enable or disable displacement degrees of freedom (DOFs) on the radiosity surface elements ([SURF251](#) / [SURF252](#)) when they are generated (equivalent to setting KEYOPT(1) after they are generated). Displacement DOFs are required to recalculate view factors at the substep level (**VFUP,DEFINE,ON**).
- **RSYMM** – Defines symmetry, rotation, or extrusion parameters for the radiosity method. Use the new *TOLER* field to increase the tolerance and compensate for a slightly offset local coordinate system when generating radiosity surface elements for planar or cyclic reflection.
- **TBFT** – Performs [material curve-fitting](#) operations. The command now supports anisotropic hyper-elastic material models and AI-based parameter initialization.
- **NLADAPTIVE** – Defines the criteria under which the mesh is refined or modified during a nonlinear solution. A new REMELM criterion is available for element removal with three options, ESTN, PSTN, and MANUAL.
- **NLMESH** – Controls remeshing in [nonlinear adaptivity](#). A new DCON control is available for automatic contact and target element generation in gaps created in the element removal process using the **NLADAPTIVE** command.

5.3. Undocumented Commands

No commands have been undocumented at this release.

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

6. Elements

This section describes changes to elements at Release 2024 R2:

6.1. New Elements

6.2. Modified Elements

6.3. Undocumented Elements

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

6.1. New Elements

The following new elements are available at this release:

- [SHELL294](#) – A 3D thermal layered shell element with both in-plane and through-thickness thermal conduction capabilities. It has no limitation on the number of interpolation layers (defined by through-thickness DOFs) whereas the corresponding legacy element, [SHELL131](#), is limited to 31 layers. In addition, it is geometrically more accurate as it distinguishes between bottom and top faces when applying surface loads, calculating view factors, and handling contact. It also considers geometry when binding adjacent elements with different layering, such as at a joint or at the runout of a tapered layer. Furthermore, it can have several material layers per interpolation layer for a better balance between accuracy and computational load. For verification examples, see [VM97](#) and [VM271](#) in the *Ansys Mechanical APDL Verification Manual*.
- [MPC184-Genb](#) – A two-node 3D element with relative rotations defined by the 3-1-3 Euler angles. Use this element to connect two bodies to transmit motion from one body to another. Stiffness and damping behavior can be specified for this joint if needed.

6.2. Modified Elements

The following elements have been enhanced:

- [CPT212](#), [CPT213](#), [CPT215](#), [CPT216](#), [CPT217](#) – These coupled pore-pressure-thermal mechanical solid elements can now be used as base elements in a reinforcing analysis using the [mesh-independent method](#).
- [FLUID30](#), [FLUID220](#), [FLUID221](#), [FLUID243](#), and [FLUID244](#) – These acoustics elements now accept harmonic nonlinear acoustic material properties BETW and SDIF, terms used in the Westervelt Equation.
- [FLUID243](#) and [FLUID244](#) – Use the new KEYOPT(2) = 8 to specify coupled nonlinear acoustic elements and KEYOPT(2) = 9 to specify uncoupled nonlinear acoustic elements.
- [PLANE222](#) and [SOLID225](#) – These coupled-field elements now support the enhanced strain or the simplified enhanced formulation (KEYOPT(6) = 2 or 3) in a structural-electric-diffusion analysis (KEYOPT(1) = 100101).
- [INTER194](#) and [INTER195](#) – The thin solid option with KEYOPT(2) = 2 (through-the-thickness, in-plane membrane, and transverse-shear deformation) now allows the use of von Mises plasticity models with [isotropic hardening](#) or [kinematic hardening](#) ([TB,CHABOCHE](#); [TB,NLISO](#) or [TB,PLASTIC](#)).
- [MASS21](#) – The mass matrix in [MASS21](#) can now be defined by inserting offset components in the element coordinate directions with KEYOPT(1) = 0 or 1 or inputting matrix constants in the element mass matrix using the new KEYOPT(1) = 2.

- [LINK228](#) – This element now supports charge-based (KEYOPT(1) = 1001) and current-based (KEYOPT(1) = 101) piezoelectric analyses.
- [TARGE169](#) and [TARGE170](#) – Now have an alternate KEYOPT(7) usage for rigid target constraints.
- [TARGE170](#) – The element's constraint type detection option (KEYOPT(5) = 0) now has values for solid-solid, shell, shell, beam-to-solid, and beam-to-shell assemblies. It can also set KEYOPT(5) to the correct value for potential over-constrained pairs and non-smooth bonding interfaces.
- [CONTA172](#), [CONTA174](#), [CONTA175](#), and [CONTA177](#) – These elements have a new KEYOPT(13) option to define the tangential contact stiffness variation (only for frictional contact).

6.3. Undocumented Elements

No elements have been undocumented at this release.

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

7. Documentation

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

7.1. Updates for Programmers

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

7.2. Feature Archive

Legacy features, commands, elements, and theory information continue to move to the [Feature Archive](#). While Ansys intends to support legacy capabilities for the immediate future, some may be undocumented in future releases. Consider moving to their recommended replacements. For information about especially notable features that have been archived, see [Features Removed or Archived \(p. 72\)](#) in the Update Guide section of this document.

8. Technology Showcase: Example Problems

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

The problems are more substantive and complex than examples found in the standard documentation set. The documentation thoroughly examines the physics involved with each problem and the consid-

erations necessary for translating problems into numerical models. Approximation issues, accuracy considerations, and recommended practices are discussed. For more information, see [What You Need to Know](#).

The following example problems have been added at this release:

- TD-64 – [Fretting Fatigue Damage with Adaptive Wear Scaling](#).

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

9. Mechanical APDL Release 2024 R2 Update Guide

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

The following topics offer supplemental 2024 R2 product-update information presented by the Mechanical APDL development and testing teams:

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

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

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

9.1. Backward Compatibility

Mechanical APDL Release 2024 R2 can read database files from all prior Mechanical APDL releases. Due to ongoing product improvements and defect resolutions, however, results obtained from old databases running in new releases may differ somewhat from those obtained previously.

9.2. Feature Updates Causing Result or Behavior Changes

The following Mechanical APDL feature updates in Release 2024 R2 are known to produce program behaviors or analysis results that differ from those of the prior release:

- [9.2.1. Change in Default PCG Preconditioner for Static and Full Transient Single-Field Structural Analyses](#)
- [9.2.2. Change in automatic constraint type detection logic for bonded contact](#)
- [9.2.3. Change in behavior when issuing LSCLEAR,ALL](#)

9.2.1. Change in Default PCG Preconditioner for Static and Full Transient Single-Field Structural Analyses

For single-field structural analyses (static and full transient), the logic which selects the default *Lev_Diff* value has been changed and will no longer automatically select a value of 1. For more information, see [PCGOPT](#).

9.2.2. Change in automatic constraint type detection logic for bonded contact

The automatic constraint type detection logic (KEYOPT(5) = 0 of target element) has been revised so that larger percentages of assembly models can be solved by default with improved robustness and accuracy. For more information, see "TARGE170 Input Summary" in the [Mechanical APDL Element Reference](#).

9.2.3. Change in behavior when issuing [LSCLEAR,ALL](#)

- Up until and including 2023R1, issuing [LSCLEAR,ALL](#) caused the component based rotational ([CMOMEGA](#)) and translational ([CMACEL](#)) inertia loads to be set to zero.
- Starting at 2023R2, [LSCLEAR,ALL](#) deletes the component based rotational and translational inertia loads. This leads to a change in behavior, in particular when using [LSWRITE](#) and [LSSOLVE](#) as previous load step loads remain active. Note that it is now consistent with the behavior of other types of loads.

9.3. Known Incompatibilities

There are no known incompatibilities introduced at this release.

9.4. Known Issues

The following issues are known to exist at Release 2024 R2:

- Instability when using the Open MPI Library and Clusters Containing AMD Architecture with a High Core Count** — When running a DMP analysis that uses the Open MPI library with clusters containing AMD architecture, the program may crash if the total core count is higher than 512. For more information on supported MPI software, see [MPI Software in the Parallel Processing Guide](#).

For issues discovered following publication of this document, see ????

9.5. Resolved Issues and Defects

The following notable issues and defects for Mechanical APDL have been resolved at Release 2024 R2:

ID	Resolution Description
928906	The echo from the CEFILTER command with CEFILTER,ON now displays the default value for the MAX NUMBER OF TERMS ALLOWED.

9.6. Features Removed or Archived

At this release, no significant features have been removed or archived.

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 enhancement has been made at Release 2024 R2:

- **Intel MPI Upgrade.** On Linux, Intel MPI support has been upgraded to version 2021.11.

Aqwa Release Notes

This release of the Aqwa related products contains all capabilities from previous releases. The following enhancements are available in release 2024 R2. Refer to the product-specific documentation for full details of the new features.

1. Aqwa Solver Modules
2. Aqwa Graphical Supervisor (AGS)
3. Hydrodynamic Analysis Systems

1. Aqwa Solver Modules

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

- **Disturbed wave elevation in Aqwa-Naut.** The time histories of the disturbed wave (including incident and diffraction wave components) elevation at the specified nodes can be calculated and output into the .plt file for further post-processing. The data can be accessed by Aqwa Graphic Supervisor (AGS) and AqwaReader.
- **Looped redundant constraints.** Looped redundant constraints are now permitted. The redundant constraint equations are automatically detected and removed; the statically indeterminate reaction forces/moments will be distributed.
- **Seabed mud layer depth.** The mud layer depth of the dynamic cables' seabed can now be defined. The default value remains as 2 meters.
- **Morison element coefficients.** Non-negative (including zero) added mass and drag coefficients are now accessible for tube, slender tube, disc, dynamic cable, and tether features. The longitudinal drag coefficients along tube and slender tube can also be defined.
- **Progress bar for large angle stability in Aqwa-Line.** A progress bar is now available to display the progressive information during a free trim large angle stability analysis in Aqwa-Line.

2. Aqwa Graphical Supervisor (AGS)

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

- **Disturbed wave elevation.** The time histories of the disturbed wave elevation records at specified nodes in the Aqwa-Naut generated .plt file can be plotted.

3. Hydrodynamic Analysis Systems

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

- **Load mapping with Morison elements.** The external surface and internal tank pressures, Morison loads, and structure accelerations calculated in a Hydrodynamic Diffraction or time domain Hydrodynamic Response analysis can be transferred to panels and line elements in a Static Structural analysis, using the Hydrodynamic Pressure Add-on, through a link on Workbench Project Schematic. The hydrodynamic forces on line bodies are mapped onto beams or pipes in the structural model; the hydrodynamic forces/moments on Morison discs are directly applied to the nearest structural mesh node. The imported acceleration may now be applied at the hydrodynamic model's COG location.
- **FMU version 2.0 and System Coupling for Aqwa-Rigid Dynamics co-simulation.** The Aqwa-Rigid Dynamics co-simulation can be run via either System Coupling or Twin Builder. FMU version 1.0 or 2.0 for the output Aqwa FMU package file can be selected based on the FMU versions supported by the co-simulation platform. System Coupling requires FMU version 2.0 for both Aqwa and Rigid Dynamics FMU package files while Twin Builder supports both FMU version 1.0 and 2.0.
- **Moonpool free surface generation.** The moonpool free surface can now be automatically created when the mesh is generated with the Moonpool Free Surface property set as **Program Controlled**.
- **Fifth order Stokes wave.** The fifth order Stokes wave can now be defined with 5 modes plus its listed properties.
- **Customized graphic properties.** The size and color for the Fixed/Connection Points and Point Mass can be customized in the Graphic Properties of each object's Details panel.
- **Non-negative added mass and drag coefficients.** Non-negative (including zero) added mass and drag coefficients are now available for line bodies, discs, dynamic cables, and tethers. The axial drag coefficient for line bodies can also be defined.
- **Seabed mud layer depth.** The mud layer depth can be optionally defined and written into the Aqwa input data files.
- **Looped redundant constraints.** Looped redundant constraint configuration is now permitted.
- **Hydrodynamic data in .CSV format.** Pressure and RAO-based nodal motions as well as internal tank pressure data can be exported to .csv files from the Hydrodynamic Diffraction results.
- **Progress bar for large angle stability.** The progress bar for Hydrodynamic Diffraction analyses now includes the large angle stability analysis status when the calculation is in session.

Ansys Composite PrepPost (ACP) Release Notes

This release of Ansys Composite PrepPost (ACP) contains all capabilities from previous releases. The following sections present the new features, enhancements, and known limitations for ACP in Release 2024 R2.

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

1. New Features and Enhancements

ACP includes new features and enhancements at Release 2024 R2 in the following areas:

- 1.1. Support for Unicode
- 1.2. Composite Failure Tool in Mechanical
- 1.3. PyDPF Composites
- 1.4. Composite Failure Criteria Migration from ACP (Post) to Mechanical
- 1.5. Sectional Properties
- 1.6. Scripting of Composite Failure Criteria in Mechanical
- 1.7. CAD Geometries
- 1.8. Enhanced Definition of Reference Directions
- 1.9. Lay-up Extrusion of Solid Models
- 1.10. Bounding Box
- 1.11. PyACP (Beta) Joining PyAnsys
- 1.12. VTK Upgrade
- 1.13. Retirement of ACP (Post)

1.1. Support for Unicode

ACP and the composite workflows in Workbench now support Unicode (UFT-8) for object names and paths. ACP can now be used in combination with most common language settings. There are some limitations in combination with Mechanical APDL that are documented in [Known Limitations in the ACP User's Guide](#).

1.2. Composite Failure Tool in Mechanical

The new **Show On Reference Surface** option is available for the result plots of the [Composite Failure Tool](#). It projects the failure values and modes of a solid model onto its reference surface. This ensures that the critical values become visible even if they are inside the solid model.

This functionality already exists in ACP (Post) and is now implemented in Mechanical because of the migration of ACP (Post) to Mechanical.

1.3. PyDPF Composites

The Python module for the postprocessing of composite structures has several improvements:

- The failure results of a solid model can be projected onto the reference surface to ensure that the critical values are visible. For a demonstration of this reference surface plot feature, see the example.
- Layered elements can now be processed independently from the preprocessing step of ACP. In this case, the section definitions are loaded from the RST file in Mechanical APDL. For more information, see the example.
- The newly added filter and averaging operators allow you to process ply-wise results. The usage of different filters and averaging strategies is shown in the example.

1.4. Composite Failure Criteria Migration from ACP (Post) to Mechanical

Composite Failure Criteria defined in ACP (Post) can now be easily transferred to Mechanical using the new Migration Helper tool. This eases the phasing out of ACP (Post) ahead of its retirement in the 2025 R1 release. For more information on the Migration Helper tool, see [ACP \(Post\) to Mechanical Migration in the ACP User's Guide](#).

1.5. Sectional Properties

Sectional properties such as elastic stiffness (EA) and torsional stiffness (GJ) are used to quantify a composite structure or to replace a shell or solid model with beam elements of equivalent properties. The computation of sectional properties is implemented in the [Section Cut](#) feature. It computes the full 6x6 stiffness matrix which can be used among others in combination with the [Preintegrated Composite Beam](#) in Mechanical APDL ([BEAM188](#) and [BEAM189](#)).

1.6. Scripting of Composite Failure Criteria in Mechanical

Composite Failure Criteria can now be created through the scripting API in Ansys Mechanical. This facilitates automated postprocessing of composite models. For more information, see [Ansys.ACT.Automation.Mechanical.Results.CompositeFailureResults Namespace in the ACT API Reference Guide](#).

1.7. CAD Geometries

The associativity of CAD Geometries in ACP is improved to better support renaming. This leads to a minor change in the scripting API, and Python scripts for ACP may need to be adjusted. Existing ACP models are upgraded automatically. For more details, see ????

1.8. Enhanced Definition of Reference Directions

The definition of the material reference directions through the [Oriented Selection Set](#) has been improved. The new property **Rotation Angle** allows you to rotate the reference directions with respect

to the orientation normal. Some examples of using this feature include defining reference directions perpendicular to an edge or converting cylindrical directions into axial directions.

1.9. Lay-up Extrusion of Solid Models

The new **Skip Elements Without Plies** option is added to the [Solid Model](#) extrusion algorithm. When this option is enabled, elements without a lay-up are automatically ignored during the extrusion. As a result, no drop-off elements are created between the elements with and without plies. You can use this option, for example, when holes are filled in the CAD geometry to improve the mesh quality but those filled areas should be ignored in the solid model.

1.10. Bounding Box

The bounding box of [Scenes](#) has been refactored to better visualize the dimensions of the model.

1.11. PyACP (Beta) Joining PyAnsys

Introducing : You can now use ACP preprocessing features as a public Python module, PyACP, in a standard Python environment outside of Workbench. PyACP can be used in combination with the PyAnsys modules PyMAPDL and PyDPF Composites, which enables the implementation of automated simulation workflows such as optimization and parameter studies. This is a beta release since some features of the standard ACP application such as Solid Model, Imported Solid Model, and Sampling Point are not available yet. For more information on PyACP, see its .

If you have any questions or feature requests, contact the composites team at composites@ansys.com or create an issue on .

1.12. VTK Upgrade

The latest version of VTK is now used in ACP, which improves visualization and performance.

1.13. Retirement of ACP (Post)

ACP (Post) is in maintenance mode and is scheduled for complete removal in the 2025 R1 release. Ansys recommends you start using Ansys Mechanical for postprocessing composite simulations. The [Composite Failure Tool](#), the [Composite Sampling Point Tool](#), and the [Envelope Solution](#) are already available in Ansys Mechanical.

For scripting, you will find a Python library for postprocess composite simulations at .

For questions or concerns, contact the composites team at composites@ansys.com.

2. Known Limitations

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

Material Designer Release Notes

This release of the Ansys Material Designer product contains all capabilities from the previous release. The following enhancement is available in Release 2024 R2. Refer to the product specific documentation for full details of the new feature.

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

1. New Features and Enhancements

Material Designer includes new features and enhancements at Release 2024 R2 in the following areas:

1.1. Short Fiber Composite Models

1.1. Short Fiber Composite Models

In the **Uniaxial Test Data** dialog, you can now directly import the experimental stress-strain curves from a file. On import, the curve is split into its elastic and plastic portions by automatically identifying the Young's modulus and yield point. The new **Experimental Data Import Plot** enables you to compare the original curve versus the imported one and review the fitting quality. For more information, see [Experimental Data \(for Short Fiber Models\) in the Material Designer User's Guide](#) and [Experimental Data in the Short Fiber Composites Guide](#).

2. Known Limitations

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

Additive Manufacturing

Additive Suite products include new features and enhancements at Release 2024 R2 in the following areas:

1. [DED Process Add-on](#)
2. [Sintering Process Add-on](#)
3. [Changes in Product Behavior - Additive in Mechanical](#)

1. DED Process Add-on

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

- You can now specify the **Number of Cooling Steps** for the cooldown portion of a simulation. This feature improves user control over viewing the part during the cooldown step, which is useful for simulations with extended cooldown times where you may want to increase the Number of Cooling Steps beyond the default of 10. This enhancement is implemented in both the DED Wizard and on the [Build Settings](#) object.

2. Sintering Process Add-on

The [Sintering Simulation Guide](#) has been enhanced to include the following advanced topics:

- [Using Location-Dependent Green Density](#): During fabrication of a part using a Binder Jet Additive Manufacturing (BJAM) or Metal Injection Molding (MIM) manufacturing method, the net-shaped powder compact may not always maintain a uniform packing/relative density. It may be beneficial to account for density variation in cases where the binder jet printer has a consistent preference for higher (or lower) compaction at certain regions within the build. Such location-dependent information can be easily implemented when modeling distortion during the sintering process.
- [Using Setters with Sintering](#): Setter plates, or setters, may be used during sintering to arrange and secure the sintered part and to prevent unwanted deformation. Learn how to include static and live setters in your sintering simulation.

3. Changes in Product Behavior - Additive in Mechanical

Mechanical 2024 R2 includes AM-related enhancements that cause product behavior to differ from earlier releases, as explained below:

- **Laser Powder Bed Fusion (LPBF):**
 - Removal steps: Contact connections between the base and the part/supports are now deleted during the base removal step. This may result in convergence improvements during [removal steps](#).

- Machine Learning Prediction [thermal strain method](#): The Machine Learning Prediction feature now requires a Structures AI+ license.
- **Directed Energy Deposition (DED)**: The code controlling element clustering for [multi-axis simulations](#) (Print Direction = 3D Beta on G-Code Clustering object) has been improved. This change may affect clustering time duration, improve convergence, and/or result in differences in results from the last release. As a reminder, the multi-axis capability is a Beta feature subject to our Beta Features Disclaimer.

Sherlock

Introduction

This document provides a summary of all new features, product enhancements, limitations, and bug fixes in the Sherlock 2024 R2 release.

1. Known Issues
2. New Features and General Enhancements
3. API Updates
4. Bug Fixes
5. Part Library

1. Known Issues

- The following issue affects Sherlock's Solder Fatigue results for stub lead components: When calculating solder joint fatigue predictions, Sherlock uses a leaded model formula based on the Euler-Bernoulli beam bending theory. Due to the geometric simplifications in the software, Sherlock models the stub leads as short beams which do not meet the assumptions in the Euler-Bernoulli beam theory.
- On Linux, when you export a trace reinforcement model and open it in Workbench, the process which assigns material properties to the mechanical model does not complete automatically. A message box will appear instructing you to create a cross-section group and then manually run the script.

2. New Features and General Enhancements

New Features

- The new Advanced Lead Meshing (ALM) feature allows you to mesh leads through their thickness and further improves lead bend meshing. You can enable ALM in the **Settings > Meshing** menu. When you perform an FEA analysis or generate FEA models with ALM enabled, Sherlock provides new lead modeling parameters so you can manage how leads are meshed. In the Parts List, you will see two new properties in the **Lead** tab—**Lead Bend Radius** and **Lead Bend Package Radius**—which define the lead bends. See [Advanced Lead Meshing in the Sherlock User's Guide](#).
- Sherlock's Mechanical Shock analysis can now use strain image maps and strain CSV files as analysis sources. See [Strain Mapping in Sherlock in the Sherlock User's Guide](#).
- You can now create a detailed component from a BGA. The detailed component is a CCA comprised of elements that are similar to those of a BGA and can be exported to Ansys Mechanical. See [Advanced BGA Modeling in the Sherlock User's Guide](#).

- When using gRPC, you can now launch Sherlock without starting the GUI by using the command-line argument **-noGUI**. See [Connecting to the Sherlock gRPC Server in the Sherlock User's Guide](#).

General Enhancements

- When running a Harmonic or Random Vibration analysis, you can now perform a single modal analysis for all selected analysis Events instead of performing a separate modal analysis for each selected Event. You can enable **Single Modal Analysis** in **Settings > FEA** menu. See [FEA Analysis Settings in the Sherlock User's Guide](#).
- The **Unused Part Property** setting in **Advanced Settings** now has three options: ENABLED, DISABLED, and HIDDEN. The new option, HIDDEN, hides from view any part properties that are not applicable to selected part. See [Advanced Settings in the Sherlock User's Guide](#).
- You can now import Sherlock trace model files (.traces) as copper layer files.
- The process that creates a CCA from a modeling region has been updated. To improve resolution, Sherlock now creates the copper layer files for the modeling region CCA as trace model files. The **Trace Mesh Size** property of the source modeling region controls the resolution of the copper trace files. See [Creating a CCA from a Modeling Region in the Sherlock User's Guide](#).
- When creating a CCA from a modeling region, Sherlock now clips the modeling region outline to that of the original CCA outline.
- Sherlock now computes the copper percentage for copper layers files much more quickly when layer snapshots are disabled.
- The **Debug Settings** panel has been improved: You can now use check boxes to select common debug tags without having to enter specific strings. See [Debug Settings in the Sherlock User's Guide](#).
- You can now apply polyline simplification to a board outline when importing ECAD archives such as ODB++ and IPC-2581. See [Importing Other File Types in the Sherlock User's Guide](#).
- You can also apply polyline simplification to a board outline from within the Outline Editor. See [Board Outline Editor in the Sherlock User's Guide](#).
- Solder Types SAC105, SAC405, Innot, and Senju M705 have been added to Sherlock's Solder Library.
- By default, Sherlock now hides the **Score**, **Overstress Score**, and **Fatigue Score** table columns for the relevant results tables.
- Previously, the only way to launch the customize-table-column feature was to right-click the Results Table. Now there is a **Customize** icon above each table and table filter.

3. API Updates

API Updates

- Added **listThermalMaps()** API to retrieve a list of thermal map files and their associated types.
- Previously, **addProject()** returned a response before processing was complete. The API will now wait until processing is finished.

- Added **updateThermalMaps()** to update thermal map files and their associated properties.
- Added **updatePartsListProperties()** which allows you to update part properties in the Parts List for a CCA. It can apply changes to multiple properties of multiple parts.
- Updated **runStrainMapAnalysis()** to support Mechanical Shock strain map analysis.
- Updated **importODBArchive()** to support polyline simplification of a board outline.
- Updated **importIPC2581Archive()** to support polyline simplification of a board outline.
- Updated **updatePartsList()** to allow for editing part properties in the Parts List
- Added the **exportTraceModel()** API.
- The **ExportTraceReinforcementModel()** message **TraceDrillHoleParam()** is being deprecated.
- Added **addThermalMaps()** API to add thermal map files and their associated properties.
- Added **exportNetList()** which exports all rows and columns of the net list table for a given project and CCA.
- Updated **addStrainMap()** API to support importing strain map images.
- Added **exportAEDB()** for exporting an Electronics Desktop EDB definition file.
- Updated the **exportFEAModel()** API to use the Advanced Lead Meshing parameters **Through Thickness Count** and **Aspect Ratio** during export if the new Advanced Lead Meshing setting is enabled. (See [New Features and General Enhancements \(p. 85\)](#).)
- Added **importProjectZipArchive()** for importing a .zip project archive when Sherlock is in multiple project mode and **importProjectZipArchiveSingleMode()** for importing a .zip project archive when Sherlock is in single project mode.

4. Bug Fixes

- Fixed a bug that arose when attempting to use the **runAnalysis()** API or the **importRstUsing()** API when importing the Assembly Tutorial project without first setting the analysis properties for the imported project.
- Resolved a bug that occurred when the **Advanced Setting** for **Unused Part Properties** is set to ENABLED: When viewing or editing a part in the Part Library, if the **Die** tab appeared disabled, the fields were still enabled. This has been corrected so the **Die** fields are disabled at the appropriate time.
- Pad part properties no longer allow invalid entries such as -1 for Pad Length.
- Resolved an issue which could cause some of the analysis to fail when running all analyses at once.
- When you double-clicked a .dfr-project file to open a project, Sherlock sometimes displayed an error that project.properties and lifeCycle.dat files were missing. The files, in fact, were not missing. This error has been corrected.

- Corrected an issue that prevented user-created BGA parts from showing up in the 3D Viewer.
- In some situations, when gRPC was in use, Sherlock would prevent a project from being deleted, indicating that one or more of the project files was in use. This bug has been resolved.
- When attempting to delete a project using gRPC, sometimes the deletion failed if nothing had been selected in the **Project Tree**. This bug has been fixed.
- Resolved an issue where the export dialogs for results tables did not allow you to select individual columns in the column list. Also, the background color of the list was corrected.
- Fixed a bug that occurred when a concave potting region with a hole cutout contained a part, resulting in an invalid geometry.
- Sherlock no longer requires mount points to be defined for an imported FE analysis.
- When parsing obsolete Gerber primitive code 22, Sherlock incorrectly defined the line with an upper-left coordinate instead of the lower-left coordinate, causing objects to be shifted. This issue has been corrected.
- Corrected an issue that prevented the File Manager from being displayed after the Advanced Filter was enabled.
- When the gRPC call **updateMechanicalShockProps()** was invoked, the **criticalStrainShock** and **criticalStrainShockUnits** properties could not be altered. This has been fixed.
- Fixed a defect that prevented the Laminate Manager advanced filtering from functioning correctly.
- Fixed a defect where arcs would have so many points that Sherlock would run out of memory.
- Potting regions that were stacked on each other did not get assigned elements to all the areas in the overlapping portions of the potting regions. This issue has been resolved.
- Updated life event editors to format values in tables to four decimal places when the absolute value is less than one. Previously, formatting was done to one decimal place. Values less than 1e-4 are formatted using three significant digits. Previously, two significant digits were used.
- When editing a node location in the Image Editor, any changes to the units in the vertex dialog repositioned the node as if it were still set to inches. This bug has been fixed.
- Fixed an issue where column filters on the Net List table were not performing case-insensitive matches.
- The Solder Fatigue analysis cycles-to-failure calculations for the BGA and CBGA were not considering balls under the die shadow when the ball pattern was set to NETLIST and the shadow ball cycles-to-failure was less than the corner ball cycles-to-failure. In this particular case, Sherlock was always using the corner ball cycles-to-failure, causing the cycles-to-failure to be higher than normal. This defect has been fixed.
- Fixed an issue with some form field labels having truncated text on multiple monitor systems with different scale settings.
- When editing parts with **Lead Geometry** set to Stub and **Lead Layout** set to Rectangular, the **Lead Long Count** and **Lead Short Count** fields were not enabled. This bug has been fixed.

- Several issues with the Japanese version of Sherlock were resolved: Removed the meaningless blank space on the **Settings > Color** pane, and translations are no longer missing on this window. Addressed the persistent problem where the **Folder Name** label on the file selection dialogs were displayed in English. And resolved the absence of frame lines on the table within the **Semiconductor Wearout** tab of the **Part Properties** window.
- Resolved the issue where the Ansys engine had an invalid .db file extension as the default for exporting data.
- When saving a Life Cycle, unsaved analysis results are no longer cleared.
- Sherlock now handles incorrect project file structures smoothly by verifying that the project directory contains both a launch file and metadata directory before adding it to the project list.
- When editing part properties, if the **Ball Pattern** is NETLIST, FULL, or UNKNOWN, the **Ball Channel Width** field gets disabled as expected.
- When editing multiple mount points, heat sinks, or mechanical parts, Sherlock now shows the length values when the values are the same, ignoring whether units are the same. Previously, Sherlock converted the values to mm when the units were different.
- When editing multiple mount points and the units were not set to mm, a dialog incorrectly indicated that the mount point was outside the board outline. This defect has been fixed.
- When creating a heat sink with units other than mm, Sherlock now sets the distance fields to the values that were saved rather than converting the saved value from mm to Heat Sink Units.
- When you are editing a heat sink, the distant values shown are no longer converted from millimeters to Heat Sink Units.
- The CTE for the solder 90Pb10Sn has been updated.
- You can now change the **File Properties** filter from Basic to Advanced without having to close and reopen the tab to affect the change.
- Exiting the application after changing the working directory no longer results in a project error.
- Sherlock no longer clips a layer image that is wider than it is tall. (Originally, the clipping was done to preserve space in the report.) However, if you wish, you can re-enable image clipping in [Settings > Report](#).
- When editing a part with a **Package Type** set to BGA, CBGA, or CGA, the **Solder** tab has now been enabled and all fields in the **Solder** tab, except for **Solder Model**, are disabled. This permits you to change the **Solder Model** for these packages while leaving non-applicable fields disabled.
- Importing results through Workbench sometimes resulted in a message that falsely reported issues were found during the import and recommended checking the Results Viewer. This bug has been resolved.
- When editing Life Cycle properties, you are no longer permitted to enter a **Duration** value and a **Number of Cycles (units = COUNT)** value that, when multiplied together, exceed the Current Service Life.
- You can now you select both a mount point and a heat sink and then select **Edit Properties**.

- Editing heat sink properties sometimes caused Sherlock to draw two heat sinks on top of each other. This issue has been fixed.
- Fixed a typo in Acceleration Factors context menu.
- In the **Add/Edit Accelerometer** dialog, Sherlock does not let you save changes if the radius is invalid, that is, less than 3 mm. However, if you changed the units to **in** or **mil** and the value was 3, Sherlock still considered the radius as invalid. This issue has been resolved. There was a similar issue in the **Add/Edit Test Points** dialog which was also resolved.
- The 3D Model Viewer is no longer displayed behind the **Edit Wire Bond Properties** dialog when it first appears.
- When editing strain images and thermal images, the displayed value of the mouse location was not always correct. This has been resolved.
- Sherlock's default memory size setting has been increased from 2G to 8G.
- Fixed the following issue: When exporting a geometry model with a part in a potting region that was completely below the base of any potting area, potting plugs were created above the part, and the potting region that covered the part was cut.
- Editing and saving a test point in the Layer Viewer no longer results in having an extra mount point in the Layer Viewer, which could be seen when moving the test point.

5. Part Library

The Part Library has been updated to include over one million parts. It is the library from 23R2.1 that has been modified. Note that the 23R2.1 library was not part of the Sherlock 23R2 release.

Here are the changes:

- Float values are correctly rounded down to three significant digits.
- Corrected formatting for 16,047 four-digit CC package names which were formatted incorrectly (for example: '402.0', '1008.0').

Forming

There is no 2024 R2 release of the Ansys Forming application.

Motion Release Notes

The following sections present the new features, enhancements and changes for the Motion Standalone Solver, Preprocessor and Postprocessor for the 2024 R2 release.

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

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

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

1. Changes in Product Behavior

Release 2024 R2 includes several new features and enhancements that result in product behaviors that differ from previous releases. These behavior changes are presented below.

- The search method for **Face Contact** detection has been changed to be the same as that for **Node Contact**.

2. New Features

The following new features are available:

- A Postprocessor function for **Contact Gap** has been added to calculate the contact gap at contact points.
- A new **User Subroutine** command has been added to establish if solver input files (such as .dfs files) were created in Workbench or the Standalone Preprocessor.
- **Rigid Body Element**, **Point Mass** and **Distributed Mass** entities are now available in the Preprocessor to create groups of nodes and additional mass entities, respectively.
- The Postprocessor now supports **multiple pages** and each page can have multiple **Animation** and **Chart** views.
- Parameters for **Twist** have been added to the **Modification** tab for **Advanced Gear Geometry** in the Preprocessor **Drivetrain Toolkit**.
- A **Signed Von-Mises** stress result has been added.

3. Functional Enhancements

The following functional enhancements are available:

- A new manual has been released detailing how to run the Motion solver from a command-line interface.
- **Maximum step size factor** and **Level of solver message** parameters have been added to the **FMI** and **M-File** export dialogs, allowing you to change the maximum step size and solver message level before co-simulation.
- The Postprocessor **Chart View** options have been improved to offer **Data Tracking** for multiple curves, along with expanded options for visibility and legend placement.
- The **View** display in the Postprocessor has been upgraded to offer multiple repositioning options.
- The method for controlling the size of the arrows for the **Vector Force** display has been changed from a sliding bar to direct numeric entry.
- The Postprocessor function for **Contact Potential Energy** has been refined to improve performance.
- Generalized **Force** and **Torque** results in a specific coordinate system due to contact forces are now available.
- **FE Film** functionality in the Preprocessor **Links Toolkit** has been improved to allow multiple types of **Contact Region** and **Pole Shape**.
- The *.dfmf file generated during the **Condensed Part** or **Imported Condensed Part** generation pass is available for both the Mechanical and Standalone environments.
- The Motion solver is available for co-simulation with Maxwell in the **Linux** environment.
- The convergence of **Modal** body **Contact** has been improved.
- **Eigenvalue Analysis** performance has been improved with faster solution and lower memory usage.
- Improved warning and information messages are now available to assist customer-model investigation when strange behavior is monitored.
- The Motion solver is now switched from **HPC** to **SMP** automatically for better performance with non-FE bodies or when an FE body has less than 500 nodes.
- The method for running the Motion solver in the **HPC** environment has been simplified and the solver now supports several options for execution.
- The **Unit** and **Remote Point** behavior information described in the Mechanical APDL *.dat input file is considered when imported into the Motion Preprocessor.
- Unexpected peak data for wheel travel vs wheel force graphs in **Half Car** analysis has been resolved.

- The selection method for characteristics, components and coordinate systems for curve drawing has been improved.

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 \(p. 99\)](#)

[CFX \(p. 117\)](#)

[CFD-Post \(p. 119\)](#)

[TurboSystem \(p. 121\)](#)

[TurboGrid \(p. 123\)](#)

[Ansys BladeModeler \(p. 125\)](#)

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

[Forte \(p. 129\)](#)

[Ansys Chemkin \(p. 131\)](#)

[FENSAP-ICE \(p. 133\)](#)

[EnSight \(p. 135\)](#)

Fluent Release Notes

The following sections contain release information for Ansys Fluent 2024 R2.

1. [Supported Platforms for Ansys Fluent 2024 R2](#)
2. [New Features in Ansys Fluent 2024 R2](#)
3. [Updates Affecting Ansys Fluent 2024 R2 Code Behavior](#)
4. [Features To Be Removed in a Future Release](#)

Backwards Compatibility: Ansys Fluent 2024 R2 can generally read case files and data files from all past Fluent releases. Solver and model settings from previous case files are typically respected. However, in some cases due to defect fixes and core improvements to improve robustness and/or performance, convergence behavior and/or results obtained may be different. Such release-to-release changes are documented in the Fluent Migration Manual, along with instructions to recover the previous behavior when possible.

1. Supported Platforms for Ansys Fluent 2024 R2

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

2. New Features in Ansys Fluent 2024 R2

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

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

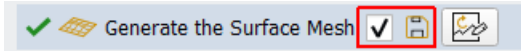
2.1. Meshing Mode

New features available in the meshing mode of Ansys Fluent 2024 R2 are listed below.

Fluent Guided Meshing Workflows

- General Improvements
 - The **2D Meshing** Workflow is now available from the list of meshing workflows for performing common meshing operations on 2D geometries. For details see [Using the 2D Meshing Guided Workflow](#).
 - The **Insert Drag Line** button has been added to the ribbon for defining custom clipping planes within the graphics window. For details see [The Ribbon](#).

- Selective checkpointing is now available for disabling or enabling the writing of back-up mesh files for individual tasks within the workflow. Checkpointing for an individual task can be enabled or disabled by hovering over the task in the **Workflow** tab.



Enabling checkpointing can be used to avoid fully restarting the workflow after clicking the **Revert and Edit** button for a task.

- The **Watertight Geometry** Workflow
 - You can now use the **Use Size Field in Solids?** option when using the poly-hexcore volume fill method to specify size-field-based sizing for polyhedra regions during volume meshing. This option is recommended when BOI(s) are defined that include polyhedra regions. For more details on using this option see [Generating the Volume Mesh](#).
 - Multiple instances of the **Set Up Periodic Boundaries** task can now be added to the workflow before and after the **Generate the Surface Mesh** task. Additionally, automatic setup of multiple translational periodic conditions can now be created within the **Set Up Periodic Boundaries** task. For details see [Setting Up Periodic Boundaries](#).
 - You can now insert the **Add Thin Volume Meshing Controls** task into the workflow to create a layered mesh within thin structures that can be combined with fill methods for volume mesh generation. In addition, this task can be utilized for meshing of stacked planar plates. For details see [Adding Thin Volume Meshing Controls](#)

2.2. Solution Mode

New features available in the solution mode of Ansys Fluent 2024 R2 are listed below. Where appropriate, references to the relevant section in the User's Guide are provided.

User Experience

- Drag and drop copying of settings in the Outline View tree is expanded and improved. Refer to [Drag and Drop, Copy and Paste in the Fluent User's Guide](#) for additional information.

Files

- You can now choose which profiles are written when writing case profiles to a file from within Fluent. Refer to [Writing Profile Files in the Fluent User's Guide](#) for additional information.
- You can now specify whether data exported to CGNS includes the family name, using the `file/export/settings/cgns-familyname` text command.
- You can now specify whether Fluent exports separate CGNS zone_t for each cell zone. This behavior is controlled using the `file/export/settings/cgns-separate-cellzones` text command.
- You can choose to append the time-step, flow-time, or iteration number to animation image files. Refer to [Animating the Solution in the Fluent User's Guide](#) for additional information.

Solver-Meshing

- The following new predefined criteria for mesh adaption are now available for aerodynamic simulations:

- **Combined Hessian Indicator**

This criterion adapts the mesh based on an error indicator that includes the Hessian—that is, the matrix of second derivatives—of several flow fields (pressure, temperature, velocity, density, and turbulence, when each is applicable). It is suitable for use across a range of Mach numbers, and can target key areas for adaption in cases with a variety of different scales and flow phenomena.

- **Mach Hessian Indicator**

This criterion adapts the mesh based on an error indicator that includes the Hessian of the Mach number. It targets adaption for a wider range of Mach number flows compared to the **Pressure Hessian Indicator**, while being based on only a single scalar value for the input field, in contrast to the multiple flow fields used in the **Combined Hessian Indicator**.

For further details, see [Aerodynamics Adaption](#).

- It is now possible to easily create multiple mesh interfaces between boundary zones that do not currently overlap. This can simplify the setup for a sliding mesh simulation in which the zones begin without overlap, and then slide into positions that include overlap. ([Manually Creating Many Non-Overlapping Mesh Interfaces](#))
- When creating non-conformal mesh interfaces, the text command for enabling adjustable tolerances (`define/mesh-interfaces/auto-options/set-one-to-one-pairing-tolerance`) is improved, so that you can manually adjust the length factor used in the pairing of boundary zones for mesh interfaces. This provides you with greater control, and may allow you to eliminate unnecessary mesh interfaces, produce missing mesh interfaces, or adjust the size of the interface areas. ([Using a Non-Conformal Mesh in Ansys Fluent](#))
- When defining an auxiliary geometry definition, a new **Reconstruction** type is available that allows you to define the geometry by building a smooth background model based on the current node coordinates of one or more specified boundary zones. ([Managing Auxiliary Geometry Definitions](#))
- It is now possible to delete cells that are marked by a specified cell register. ([Deleting Cells](#))
- When defining a rigid body zone as part of a dynamic mesh simulation, you can now use a motion definition to specify the motion for the zone. A motion definition is comprised of a name, reference frame, translation definition, and rotation definition. Using a motion definition may be advantageous, as it can be simpler to set up than the alternatives (that is, a profile or user-defined function) and allows you to use expressions to define the dynamic mesh motion. ([Managing Motion Definitions](#))

Cell Zones and Boundary Conditions

- For cases with solid zone(s) that involve perforated wall boundary conditions, you can now include the effect of conjugate heat transfer between the solid walls and the flow when the injection flow passes through the perforated holes.

- For a case that has a solid cell zone that involves solid motion with the energy equation enabled, you can now enable or disable the modeling of boundary advection at the individual boundary wall zones that are adjacent to the solid zone. Such modeling accounts for the heat that advects into or out of the domain because of the solid motion. By default it is disabled everywhere. In previous releases such boundary advection was by default applied to all of the boundary walls when the solid motion was translational and to none of the walls when the solid motion was rotational. It is now recommended that boundary advection be enabled at boundary walls where the average velocity of the solid is normal to the wall to a significant degree, and only then; this applies for translational as well as rotational motion. When you read a case created in a previous release, Fluent will automatically revise the boundary advection settings to match the new recommended practice; you can undo such changes using the `define/boundary-conditions/wall` text command. ([Boundary Advection for Solid Motion](#))

Materials

- The REFPROP v9.1 database previously used by the NIST real gas models has been replaced by the REFPROP v10.0 database containing 152 fluids. ([The REFPROP v10.0 Database](#))

The REFPROP v7.0 database is no longer supported and is not included with the Ansys Fluent installation.

Heat Transfer / Radiation

- For the GPU solver, the Discrete Ordinates (DO) radiation model is now available.

Turbulence

- For the GPU solver, the Generic WMLES model is now available ([Features Supported by the Fluent GPU Solver](#)).

Turbomachinery

- When modeling a large pitch difference with an NPS interface, you can now minimize total pressure losses across the interface using the following text command: `define/turbo-model/general-turbo-interface-settings/expert/nps-minimize-po-loss`. ([No Pitch-Scale interface](#))
- A turbomachinery performance tool is now available to evaluate various efficiencies and performance characteristics of a turbomachine. ([Calculating Turbomachine Performance](#))

Reacting Flows

- When modeling multicomponent particles with chemical reactions, you can now specify generic values of viscosity and surface tension for the liquid urea and its by-products using the `define/models/dpm/options/scr-urea-deposition/set-liquid-properties` text command. ([Multicomponent Particles with Chemical Reactions](#))

Discrete Phase Model

- When modeling multicomponent particles with chemical reactions, the **Viscosity**, **Thermal Conductivity**, and **Droplet Surface Tension** properties of a particle-mixture material are no longer limited to constant.

- For cases with multicomponent particles, **immiscible-not-vaporizing** has been renamed as **none** in the **Evaporating Species** selection list in the **Set Injection Properties** dialog box (**Components** tab).
- For cases that involve multicomponent particles with chemical reactions, you can now model thermolysis reactions in free-stream particles.
- The heat transfer coefficient models, which were previously available only for cases with inert or combusting particles, are now made available also for DPM cases that include multicomponent particles that do not contain evaporating species.
- For cases that involve the tabulated particle diameter distribution, you can now use the **Interpolate Between Classes** option to generate particle diameters by linear interpolation between neighboring diameter classes. ([Using the Tabulated \(Discrete\) Diameter Distribution](#))
- For simulations that involve both evaporating liquid droplets and Lagrangian wall film, summary reports of evaporated mass are now fully supported. Using this functionality, you can include information about the fractions of total vapor from free-stream particles and from evaporating film in an extended summary report. This was available as a beta feature in Ansys Fluent version 2024 R1. The report/evap-mass-details-in-dpm-summ-rep text command has been renamed as report/calc-exchange-data-on-zone-types. ([Evaporated Mass](#))

Multiphase Models

- For the two-resistance heat transfer model, the option formerly referred to as the **lavieville-et-al** interphase heat transfer option in the **Two Resistance Model** dialog box is now renamed as the **time-scale** option for clarity. In addition, you can now modify the time scale value.
- For Eulerian multiphase cases that involve the boiling model, you can now modify the bubble contact angle for the **kocamustafaogullari-ishii** bubble departure diameter model.
- The phasic wall heat fluxes for the VOF and Mixture multiphase models can now be calculated using a new approach which makes use of the linearized coefficients at the phase level. These coefficients are then accumulated at the mixture domain for correct reporting of wall heat fluxes at the wall boundaries. ([Heat Transfer and Radiative Flux Distribution for Non-Eulerian Multiphase Models](#))
- A new postprocessing field variable, **Saturation Vapor Pressure n** , is now available in the **Phase Interaction...** category.
- For multiphase cases that involve the cavitation mass-transfer model, a new postprocessing field variable, **Saturation Vapor Pressure n** , is now available in the **Phase Interaction...** category.

Electric Potential Field and Electrochemistry Model

- For electrolysis and H2 pump cases, a new postprocessing field variable, **Capillary Pressure**, is now available in the **Potential...** category.
- For the electrolysis and H2 pump model, the following user-defined functions have been added:
 - **DEFINE_ELECTROLYSIS_ECHEM_RATE**: To customize transfer currents in the Butler-Volmer equations.

- **DEFINE_ELECTROLYSIS_RELATIVE_PERMEABILITY**: To customize the relative permeability used in the transport equation of the volume fraction of liquid water.

See **DEFINE_ELECTROLYSIS_ECHEM_RATE** and **DEFINE_ELECTROLYSIS_RELATIVE_PERMEABILITY** in the *Fluent Customization Manual* for details.

- The **DEFINE_CAPILLARY_PRESSURE** user-defined function has an extra argument—liquid saturation *S*. An example has been added to show the usage of the new argument. The existing **DEFINE_CAPILLARY_PRESSURE** UDFs must be updated accordingly. (**DEFINE_CAPILLARY_PRESSURE**)
- When using the electrolysis and H2 pump model, you can now specify total voltage and total current of a cell or stack using an expression.
- When modeling unresolved electrolysis, you can now specify the electrical conductivity for the anode and cathode catalyst layers and consider the ohmic potential loss.

In addition, for clarity, the following parameters have been renamed and moved to different tabs in the **Potential/Electrochemistry** dialog box:

Old Name	New Name	Old Tab	New Tab
Anode Catalyst Surface Volume Ratio	Surface Volume Ratio	Electrolyte	Anode
Cathode Catalyst Surface Volume Ratio	Surface Volume Ratio	Electrolyte	Cathode
Anode Catalyst Thickness	Layer Thickness	Electrolyte	Anode
Cathode Catalyst Thickness	Layer Thickness	Electrolyte	Cathode

Battery Model

- A new model, the battery venting model, is now available for simulating some battery hazard conditions. ([Battery Venting Model in the Fluent Theory Guide](#))
- For the standalone thermal abuse model, the following enhancements have been introduced ([Standalone Thermal Abuse Model](#)):
 - You can now plot the **Heating Temperature vs Temperature Profile**. The **Heating Rate Profile** plot has been renamed to **Heating Rate vs Time Profile**.
 - You can now compare the temperature prediction with imported experimental data.
- The battery model can now be coupled with the multiphase model. This can be used, for example, in simulations of immersion cooling of a battery system in certain battery thermal management applications.

Polymer Electrolyte Membrane Fuel Cell (PEMFC) Model

- For transient simulations of PEM fuel cells, the ice formation model has been added. The model can predict the formation of ice inside the PEMFC device when it is started under freezing conditions. ([The Ice Phase Model](#))
- You can now specify total voltage and total current of a cell or stack using an expression.

Fluent Native GPU Solver

- The Fluent GPU Solver can now be run on AMD GPUs.
- The following capabilities of Fluent solution mode are now supported by the Fluent GPU Solver:
 - Compressible flow at mass flow inlets and outlets.
 - Compressible flow at velocity inlets.
 - Pressure far-field boundaries.
 - Low speed compressible flow.
 - Solution controls for Pseudo time method and flow courant number.
 - Data sampling for steady statistics.
 - Conjugate heat transfer (CHT) with anisotropic and orthotropic thermal conductivity for solids.
 - Species transport with the eddy-dissipation model for turbulence-chemistry interaction.
 - Surface report definitions:
 - Standard Deviation
 - Volume Flow Rate
 - Uniformity Index - Mass Weighted
 - Volume report definitions
 - Volume
 - Mass
 - Poor mesh numerics (poor cell removal).

Density-based Solver

- A linear ramp-up of Courant number is now available for Solution Steering. ([Using Solution Steering](#))

Adjoint Solver Module

- In previous releases, when using an expression as an adjoint observable, only single-valued scalar expressions were supported. In this release, support for combined single-value and field expressions within a binary operator has been added for defining expressions as observables.
- The **Goal-Based Error Indicator** predefined criteria is now available for adapting the mesh to reduce the influence of mesh refinement on an observable as part of an adjoint calculation.

Graphics, Reporting, and Postprocessing

- You can now control the tessellation of 3D vectors in vector plots. Reducing the tessellation value leads to faster vector rendering, at the expense of the appearance of the arrows.
- You can easily measure distances and angles of objects displayed in the graphics window. Refer to [Measuring Distance and Angle in the *Fluent User's Guide*](#) for additional information.
- You can now visualize profile fields as a cloud of points. Refer to [Viewing Profile Data in the *Fluent User's Guide*](#) for additional information.
- Annotations are improved to allow for the appending of date, time, flow time, time step, and more. You can also move annotations around after they are displayed. Refer to [Adding Text to the Graphics Window in the *Fluent User's Guide*](#) for additional information on creating annotations.
- The new [Figure 1: Learning and Support Ribbon Tab \(p. 106\)](#) provides quick access to tutorials, training resources, documentation, and more.

Figure 1: Learning and Support Ribbon Tab



- You can now directly select dashboards as an animation object from within the **Animation Definition** dialog box. Refer to [Animating Embedded Window Dashboards in the *Fluent User's Guide*](#) for additional information.
- Dashboards now support embedding plots and graphics objects without specifying a base window, thus allowing you to tile your windows such that there is no overlap. Refer to [Embedded Graphics Window Dashboards in the *Fluent User's Guide*](#) for additional information.
- Realistic raytracing displays now look even better with the ability to include the shadows cast by the displayed model on the environment ground. You can also lock the model's location in the environment to fine tune its appearance. Refer to [Realistic Rendering Using Raytracing in the *Fluent User's Guide*](#) for additional information.

Parallel Processing

- When running on Linux, a new ethernet .efa interconnect is available for Amazon Web Service (AWS) instances with the EFA interface. ([Starting Parallel Ansys Fluent Using Fluent Launcher](#))

Fluent for Arm

- The following features are now supported when running Fluent for Arm:
 - WebUI
 - Turbo acoustics where the FFT fields are computed using the full signals

For more details, see [Running Ansys Fluent on Arm Compute Nodes](#).

Parametric Studies

- You can now make use of GPU resources for concurrent design point updates defined locally. See [Running Locally Concurrent Design Point Updates with GPUs](#) for more information.
- optiSLang-related features have been moved to their own category and expanded upon in the Parametric Ribbon. Now, the optiSLang **Capabilities** portion of the ribbon contains **Data Export** options for exporting data, directly opening optiSLang, or its postprocessor for further design point analysis. See [Exporting Parametric Designs to optiSLang](#) for details.
- When defining design points for use with optiSLang, you now have more options to optimize, customize, and run your designs of experiment (DOE). See [Creating and Optimizing Designs of Experiments for optiSLang](#) for details.

Simulation Reports

- Simulation reports exported to Powerpoint presentations now include breadcrumbs and descriptions for individual graphics (such as contours, plots, etc.).

Cell Registers

- When creating a cell register based on a field variable, a new **Hessian** approach is available from the **Derivative Option** drop-down list. This approach allows you to calculate an error indicator that includes the Hessian (the matrix of second derivatives) of the field variable, which can be useful when creating a cell register to use as part of mesh adaption for an external aerodynamic simulation. ([Approaches For Deriving Field Values](#))

Expressions

- You can now change the units of in-use named expressions.

Load Managers

- The **Scheduler** tab in Fluent Launcher is improved to provide the following options (which were previously only available as command line options):
 - You can now specify a submission host when running under LSF.
 - When running under Slurm, you can now specify the number of node processes per cluster node and, if you have enabled the **Native GPU Solver** option in the **Home** tab, you can now specify the number of graphics processing units (GPUs) per cluster node.

For further details, see [Running Fluent Using a Load Manager](#).

Web Interface

- To better align with the standard desktop version of Fluent, the following enhancements have been introduced to the web interface:
 - You can now select multiple items in the Outline View by holding down the **Ctrl** key while you make your selection. Context menu options are available when applicable.

- Drop-down menu options now appear properly capitalized.
- You can now create and view plots for your named expressions.
- Listings and categorizations of report definitions and surfaces in the Outline View have been simplified.
- Improvements have been made to Fluent's behavior when interrupting or pausing a batched/journaled simulation while the web server is running, where Fluent would have previously exited upon interrupting a journal, now a timeout period is provided.
- At the top level of the Outline View, cell zone and boundary condition listings can be displayed in various ways using the **Group By** context menu option where you have the choice of displaying a **List View**, by **Name**, by **Adjacency** (for boundaries only), or by **Zone Type** (the default).
- In addition to the Solution and Results aspects of your Fluent session, you now have access to the Setup of your simulation where you can interact with available simulation settings, model options, cell zone, boundary conditions, and so on. See [Interacting with Your Simulation Setup](#) for more information.
- You can now interactively move and resize the color map for results-based plots such as contours, vectors, etc.
- You can now edit multiple objects at the same time (especially useful for boundary conditions) using the Outline View and the context menu. See [The Graphics Window](#) for details.
- You can now create mirror planes using the web interface. This allows you to have a fully represent a symmetrical model by mirroring off of the symmetry plane. See [Creating and Displaying Mirror Plane Objects](#) for details.
- You can now interactively use the mouse to probe your results and visualize specific data values in specific spots. This is available by using the corresponding mode in the Visualization Arc and clicking a specific spot on the plot in the graphics window. See [Editing Multiple Objects at Once](#) for details.
- You can now select certain objects (report definitions, contours plots, etc.) in the Outline View tree and copy them to the system clipboard (using the corresponding context menu option). You can subsequently paste them into another selected object of the same type in the tree (using the corresponding context menu option). You can copy/paste such objects within the same session, or between different sessions. See [The Outline View](#) for more information.
- There is a new **Theme** preference that allows you to choose between the default **dark** color theme or a **light** color theme for the web interface.
- Plots have been improved to include:
 - The ability to resize the window once it has been relocated outside of its docked position (see [The Plots View](#) for details).
 - Enhancements for layout selection for multiple plots (see [Displaying Multiple Plots](#) for details).
 - The ability to create cumulative plots using the web interface.

- The ability to plot histogram data, rather than just the ability to print data to the console.

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 2024 R2 Beta Features Manual*.

2.3. Fluent Workspace Applications

New features available in the client applications of Ansys Fluent 2024 R2 are listed below.

Fluent Icing

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

- **Impact ice density model**

The impact ice density model which estimates the ice density based on local droplet impact angle and freezing fraction is upgraded to release, introducing three parameters that can be used to tune the model. These parameters are the maximum and minimum rime ice densities, and the blending function coefficient that controls the transition between the two based on the impact angle. A special handling of runback ice that forms past impingement limits where the impact angle is 0 is introduced.

- **Icing with adiabatic wall boundary conditions when using particle size distributions**

Use of adiabatic walls requires running EID and vapor simulations before ice computations. Adiabatic wall option was limited to monodispersed droplet multishot icing simulations in the earlier versions, due to the unavailability of the combined vapor solution file with particle size distribution runs. This limitation is lifted by providing combined vapor solution data alongside combined droplet and crystal solution fields. To facilitate the combination of different bins with vapor, the vapor fields are now saved in droplet and crystal files and combined with the rest of the data in these files. As part of this update, vapor solution files will no longer be visible in the project view if using the Continuous (DROP3D) particle solver. Ice step will use vapor fields directly from droplet or crystal solution files. Vapor files will only be written out if running vapor only. This update enables using adiabatic walls when running with particle size distributions, since adiabatic walls require EID and EID requires vapor solution.

- **De-icing enhancements**

- The unsteady de-icing feature can now display transient ice shapes.
- Inner solver iterations are added to de-icing CHT time steps to converge the boundary conditions better, synchronizing air and ice side of temperatures and heat fluxes.
- De-icing runs can now use a restart ice solution, to permit scenarios like running with heat-off for a long duration to accumulate ice, then use that as a restart to test various de-icing schemes.

- **Anti-icing updates**

The anti-icing workflow will now start with the dry air solution to extract the HTC and surface recovery temperature. EID will be used for the HTC computation

- **CFX import (beta)**

CFX .res files can be imported using the New Simulation button, similar to importing a .cas file. The mesh, the flow solution, and a limited array of settings will be imported, to be used as the air solution for particle and icing calculations. Some material properties (Cp, etc.) normally supplied as CVS files to CFX can be imported as well. This is currently a beta feature.

Fluent Aero

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

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

- **Parametric Post-Processing**

- The parametric post-processing feature, initially introduced as a Beta feature in 2024 R1, has been upgraded to full release in 2024 R2. It is now the default method to post-process Fluent Aero solutions and is accessible from the Results node in the Outline View. This upgrade brings several enhancements aimed at improving user experience and efficiency in analyzing CFD results. These include an easier view management system, customizable image saving options, and the ability to save all the graphics results for all the design points and to create animations with a single click.

- **Aerodynamic Extraction Tool (AET)**

- The *Aerodynamic Extraction Tool* (AET), an automated workflow intended to extract the performance characteristics of a rotor blade, and to produce airfoil performance definition files that can be used to define the rotor performance in Fluent *Virtual Blade Model* (VBM) has been improved to better support large twist angles of blade sections.

- **Custom Launch Script**

- A Custom Launch Script can now be used to launch the Fluent Solver connected to a simulation in Fluent Aero. This allows for complete flexibility to specify any launching option (cluster, solver, etc.) as required by a particular computing machine or in more complex computing environments.

- **Reynolds Number Criterion**

- Users can now set the Viscous Model to Reynolds Number Based in the Airflow Physics panel. This option allows the specification of two Reynolds Number Thresholds, which define the three distinct flow regimes: Laminar, Transition and Fully Turbulent. A laminar flow model will be used to solve any design point in the Laminar flow regime. For design points that fall within the Transition and Fully Turbulent flow regimes, two different viscous models can be specified. In this manner, appropriate viscous models can be specified at each design point based on farfield or wind tunnel inlet Reynolds numbers.

- **Turbulence Models**

- The Spalart-Allmaras turbulence model has been added as a selectable Viscous model in Fluent Aero.
- Corner Flow Correction has been added as a selectable option for all Viscous models available in Fluent Aero that support this option.

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

- The Parameter Search feature has been expanded to search for Stall Angles of Attack associated with a particular geometry and flight condition. This search uses a hybrid methodology to identify the maximum Lift Coefficient within a range of angles of attack. This hybrid methodology applies first a Golden section method to identify a narrowed search interval, and then a quadratic interpolation method to speed-up the search of the local maximum within this narrowed interval. The Stall AoA can be subsequently used as an input to STK Aviator when defining a particular aircraft's performance.

- **Mesh Adaption (Beta)**

- A new **Adaption** node is available under Solution in the Outline View of Fluent Aero. This feature automatically adapts the baseline mesh of the simulation using the polyhedral unstructured mesh adaption (PUMA) in combination with the Combined Hessian Indicator. Mesh adaption cycles are used to progressively refine the baseline mesh in order to reveal the most salient flow features, shocks, wakes, etc. of your simulation, therefore improving the precision of CFD calculations. Fluent Aero's mesh adaption is supported across multiple design points, where each design point starts with the same baseline mesh and is subsequently adapted based on that particular design point's conditions and solution. This feature helps users to achieve accurate mesh independent solutions that capture detailed flow phenomena that may not have been possible using the baseline mesh over a wide range of conditions.

See [Fluent Materials Processing Workspace](#) for more information.

3. Updates Affecting Ansys Fluent 2024 R2 Code Behavior

The following sections list the code changes in Ansys Fluent 2024 R2.

- 3.1. [Meshing Mode](#)

- 3.2. [Solution Mode](#)

- 3.3. [Fluent Workspace Applications](#)

General

3.1. Meshing Mode

This section contains a list of code changes implemented in the meshing mode of Ansys Fluent 2024 R2 that may cause behavior and/or output that is different from the previous release.

Meshing Workflows

- General Improvements
 - Minor changes have been made to polyhedra mesh behavior which will generally improve quality. This may result in some re-balancing of total cell count, with more poly-prisms being generated and fewer polys. Overall, the total cell count remains similar to that of previous releases.
- The **Watertight Geometry** Workflow
 - In previous releases, in the **Generate the Volume Mesh** task, the **Normal Smooth Relaxation Factor** was set to 1 by default. In this release, the normal smooth relaxation factor has been set to 100 by default, resulting in improved default prism behavior.
- The **Fault-Tolerant** Workflow
 - In previous releases, in the **Generate the Volume Mesh** task, the **Prism Normal Smooth Relaxation Factor** was set to 1 by default. In this release, the prism normal smooth relaxation factor has been set to 100 by default, resulting in improved default prism behavior.

3.2. Solution Mode

This section contains a list of code changes implemented in the solution mode of Ansys Fluent 2024 R2 that may cause behavior and/or results that are different from the previous release.

Materials

- The reactions of the urea-water-deposits-air-brack mixture material in the Ansys Fluent materials database have changed to use an updated Brack reaction mechanism for urea deposits chemistry, which provides closer agreement to experimental urea decomposition curves. As a result, you may see improvements in solution accuracy of simulations that involve multicomponent particles with chemical reactions compared with previous releases.
- In the Fluent materials database, the piecewise polynomial profile of the air specific heat in the temperature range of 1000 K to 5000 K has been corrected. This will produce more accurate results for simulations in high temperature ranges.

Turbomachinery

- An enhanced version of the mixing-plane discretization has been implemented to prevent turbulence quantities from hitting limits in certain flow conditions. Typically, you may not notice major differences in results between this and prior versions. However, you can revert to the previous implementation using the following text command:

```
define/turbo-model/general-turbo-interface-settings/expert/backward-
compatibility/pre-24r2-mp-discretization
```

Turbulence

- When using the standard roughness model in combination with the following turbulence models:
 - two-equation models based on the omega-equation,
 - standard, RNG, and realizable k-epsilon models, when they are used with standard and scalable wall functions,
 - Reynolds stress models, when they are used with standard and scalable wall functions,

the extrapolation from cell center to the wall face has been corrected and may create slight difference in the solution for velocity, temperature, omega, and epsilon.

Discrete Phase Model

- For cases that involve wet combusting particles together with the Lagrangian wall film model, the calculation of the wall-film particle thermal conductivity has been improved to account for both combusting and droplet particle materials. The previous implementation accounted only for the droplet particle material. As a result, the prediction of the wall film temperature and film boiling rate should be more accurate compared to previous releases.
- For cases with source term linearization, the enhanced source-term linearization method for the discrete phase source terms is now enabled by default. The method generally improves solution convergence, and, as a result, the convergence behavior may change slightly compared to previous releases. Settings in existing cases are respected. If you want to revert to the standard linearization method, you can issue the following text command:

```
define/models/dpm/interaction/enhanced-source-term-linearization-
enabled? No
```

Multiphase Models

- For cases that involve the Eulerian multiphase boiling model with either the **Heat Flux** or **Coupled** thermal boundary conditions at walls, the algorithm for calculating wall temperatures has been improved. As a result, the temperature distributions on walls will be more accurate compared to previous releases.
- The bisection method, which is used to determine the wall surface temperature for partitioning the wall heat flux in Eulerian multiphase boiling simulations that involve the heat flux wall boundary conditions, has been improved. It now incorporates a more accurate algorithm for generating initial guesses for the initial minimum and maximum values of the wall surface temperature range. This will result in more accurate solutions for some cases, such as those with relatively high wall heat flux.

Field Variables

- The **Cell Wall Distance** field variable (in the **Mesh...** category) is no longer available if the case file does not contain any walls.

Parallel Processing

- For Windows with more than 64 logical cores, the default MPI is now changed to Microsoft MPI. The default remains Intel MPI for all other cases (that is, Windows with less than 64 logical cores and Linux). To specify an MPI, you can use the **MPI Types** drop-down list in the **Parallel Settings** tab of Fluent Launcher or use the `-mpi=` command line option.

User-Defined Functions (UDFs)

- The `CX_Message` utility is no longer supported and should not be used as it may cause the Ansys Fluent session to hang infinitely. Instead of `CX_Message`, use the `Message` utility for printing data to the console. See [Message](#) for details.
- A new argument, `const char *solver`, has been added to many Fluent built-in functions. Typical examples of the affected functions are `Scalar_Derivatives(...)` and `Scalar_Reconstruction(...)`. To update your user-defined function (UDF), follow these steps:
 1. For any Fluent built-in functions used in your UDF, check to see if the argument list has changed and update the UDF source code accordingly. Use the `"rp-global"` string constant (in double quotes) as an argument for the new call parameter (unless you are instructed otherwise).
 2. When compiling your UDF, carefully check and address all compiler warnings.

3.3. Fluent Workspace Applications

This section contains a list of code changes implemented in the client applications of Ansys Fluent 2024 R2 that may cause behavior and/or output that is different from the previous release.

Fluent Materials Processing

- – 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 Fluent Materials Processing 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 2024 R2, but will be removed in a future release of Ansys Fluent. It is recommended that you begin to migrate any cases that use these features at your earliest convenience.

Fluent as a Server Session Management

- **Fluent as a Server**
 - This feature is deprecated and will no longer be supported after release 2024 R2. You should instead migrate to using PyFluent (<https://fluent.docs.pyansys.com/>). All documentation regard-

ing this feature has been removed as of release 2024 R1, however, you can find the feature documentation in previous releases.

Solution Mode

• **Files**

- Import of the following file types is **Removed**:
 - FIDAP
 - I-deas Universal
 - LSTC
 - Marc POST
 - PATRAN Neutral
 - PTC Mechanical Design
 - Fluent 4 Case File
 - PreBFC File
- Export of the following file types is **Removed**:
 - AVS
 - Data Explorer
 - I-deas Universal
 - PATRAN Neutral
 - PATRAN Nodal (was TUI only)
- Import and export of the following Fluent-Solid Interaction (FSI) file formats is **Removed**:
 - I-deas Universal
 - Mechanical APDL
 - PATRAN Neutral

Note:

If you still require access to importing/exporting any of the above mentioned formats, enter (enable-feature 'deprecated-import-export-formats) in the console.

- **Parallel Processing**

- Using General Purpose Graphical Processing Units (GPGPUs) in offload mode to accelerate the calculations of the following is deprecated, and will be removed in a future release. Note that offload mode (which involves the **Solver GPGPUs per Machine (Offload Mode)** setting in Fluent Launcher, the `-gpgpu` command line option, and/or specialized text commands, as described in the links that follow) is distinct from the Fluent native GPU solver. It is recommended that you migrate to using the native GPU solver, using the fully supported features where possible (see [Using the Fluent Native GPU Solver](#)) or trying the beta features where necessary (see [Fluent GPU Solver Beta Features in the Fluent Beta Features Manual](#)).
- accelerated Algebraic Multigrid (AMG) solver ([Using General Purpose Graphics Processing Units \(GPGPUs\) With the Algebraic Multigrid \(AMG\) Solver](#))
- accelerated view factor calculations (using the hemicube or ray tracing method) for the surface-to-surface radiation model ([Accelerating View Factor Calculations Using General Purpose Graphics Processing Units \(GPGPUs\)](#))
- accelerated Discrete Ordinates (DO) radiation model calculations ([Accelerating Discrete Ordinates \(DO\) Radiation Calculations](#))

Fluent Workspaces

- The Fluent Remote Visualization Client is deprecated. As an alternative you can try [Remotely Accessing Your Simulations Using Ansys Fluent's Web Interface in the Fluent User's Guide](#).

CFX Release Notes

The following sections contain release information for Release 2024 R2 of Ansys CFX.

- 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 2024 R2 of Ansys CFX.

- For injection regions having **Injection Model** > **Option** set to **User Defined Slot** or **Cylindrical Slot**, the approximate shape of the slot is outlined in the **3D Viewer** of CFX-Pre. Other injection regions continue to be rendered as symbols. For details, see [Injection Regions in the CFX-Pre User's Guide](#).
- Two start methods have been replaced by newer start methods as follows:
 - 'Submit to UGE Queue' has been replaced by 'Submit to UGE'.
 - 'Submit to SLURM Queue' has been replaced by 'Submit to Slurm'.

The old start methods are still available but might be removed in a future release.

Benefits of the new start methods include:

- The job scheduler command is defined in the new start methods rather than being hard-coded in the script code, allowing easier customization.
- With the new start methods, new job submission command line options can be used with the `cfx5solve` command. For details, see [Command-Line Options and Keywords for cfx5solve in the CFX-Solver Manager User's Guide](#).
- For eligible cases, there are **Stabilization Control** settings to automatically reduce the timestep size or time scale value and reattempt any timesteps/iterations that fail to converge or that exhibit solution divergence. For details, see [Stabilization Control in the CFX-Pre User's Guide](#).
- For steady-state cases and harmonic balance cases, you can define *Dynamic Additional Variables* (or simply *Dynamic Variables*) and then use them to change expression-based settings gradually, so as to maintain solution stability during convergence. For details, see [Dynamic Additional Variables in the CFX-Solver Modeling Guide](#).

- New modeling options are available for particle induced turbulence. For details, see [Enhanced Turbulence Production Models in the CFX-Solver Modeling Guide](#).
- The Marie wall function provides special treatment for turbulence at walls to improve solution accuracy for multiphase flow. For details, see [Shear Velocity Scaling Model in the CFX-Solver Modeling Guide](#).
- For directly specified interphase mass transfer sources, you may specify a fluid-specific linearization coefficient for mass flux or flow. For details, see [Mass Transfer in the CFX-Pre User's Guide](#).

Two new settings, **ipmt false timestep option** and **ipmt false timestep multiplier**, add control of the linearization of terms in equations for interphase mass transfer. For details, see [Discretization Parameters in the CFX-Solver Modeling Guide](#).

CFD-Post Release Notes

The following sections contain release information for Release 2024 R2 of Ansys CFD-Post.

1. Supported Platforms
2. New Features and Enhancements
3. Incompatibilities
4. Updates Affecting Code Behavior

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".
-

2. New Features and Enhancements

This section lists features and enhancements that are new in Release 2024 R2 of Ansys CFD-Post.

- The rendering of an injection region can now be, and is by default, clipped at a certain (adjustable) value of its associated area fraction variable. The result is a better visual representation of the shapes of the holes/slots. For details, see the description for the **Use Post Region Rendering** check box and **Area Fraction** setting in [Injection Regions in the CFD-Post 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 2024 R1), and the support changes (functionality that is no longer supported) in Release 2024 R2 of Ansys CFD-Post.

- When used in a quantitative calculation in CFD-Post, an injection region locator now applies strictly to the hole/slot portion of an injection region; such a locator no longer includes any mesh faces or parts of mesh faces that do not overlap a hole/slot. As a result, quantitative calculations in CFD-Post that involve an injection region locator's area should not normally require reference to the injection region's associated area fraction variable.

4. Updates Affecting Code Behavior

This section contains a list of changes that may cause the computed results from Ansys CFD-Post to differ between Release 2024 R2 and Release 2024 R1.

- As mentioned earlier, quantitative calculations in CFD-Post that involve an injection region locator's area should not normally require reference to the injection region's associated area fraction variable. Consequently, CFD-Post session (*.cse) and state (*.cst) files might need to be modified to avoid such reference.

TurboSystem Release Notes

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

These release notes cover:

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

These release notes do not cover:

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

Note:

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

1. Supported Platforms

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

2. New Features and Enhancements

This section lists features and enhancements that are new in Release 2024 R2 of TurboSystem.

- The Turbomachine Neutral Data Format now supports cut-off/square leading/trailing edges for blades designed using profile points. For details, see [Appendix 2: BNF Rule List in the *TurboSystem User's Guide*](#).

TurboGrid Release Notes

The following sections contain release information for Release 2024 R2 of Ansys TurboGrid.

1. [Supported Platforms](#)

2. [New Features and Enhancements](#)

1. Supported Platforms

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

2. New Features and Enhancements

This section lists features and enhancements that are new in Release 2024 R2 of Ansys TurboGrid.

- The new settings **Hub High-fidelity Features Cross Periodic Boundaries** and **Shroud High-fidelity Features Cross Periodic Boundaries** allow high-fidelity hub/shroud/blade geometry to cross the periodic boundaries of the passage at locations not on the low-fidelity hub/shroud. For details, see [Complex Blade End Tab in the TurboGrid User's Guide](#).
- The TurboGrid system's Turbo Mesh cell has a new property, Solution Process > Update Option, which enables you to choose whether updating the cell is done in the foreground or background. For details, see [TurboGrid in the TurboSystem User's Guide](#).
- A new preference, **ATM Five Star Vertex Mesh Size Factor**, has been added that affects mesh smoothness. For details, see [TurboGrid Options in the TurboGrid User's Guide](#).
- In stand-alone mode, when opening a TurboGrid initialization file that contains secondary flowpaths, you can select which, if any, secondary flowpaths to load.

Ansys BladeModeler Release Notes

The following sections contain release information for Release 2024 R2 of Ansys BladeGen, Ansys BladeEditor, and Ansys BladeBuilder.

1. [Supported Platforms](#)
2. [BladeGen](#)
3. [BladeEditor](#)
4. [BladeBuilder](#)

1. Supported Platforms

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

2. BladeGen

2.1. New Features and Enhancements

This section lists features and enhancements that are new in Release 2024 R2 of Ansys BladeGen.

Release 2024 R2 of BladeGen has no new features or enhancements.

3. BladeEditor

3.1. New Features and Enhancements

This section lists features and enhancements that are new in Release 2024 R2 of Ansys BladeEditor.

- The **Export BladeEditor NDF file > Profile Point Mode** command now supports:
 - Leading and/or trailing edges of type "Cut Off" or "Square".
 - Splitter blades

For details, see [Exporting Currently Existing Blades to NDF Files in the TurboSystem User's Guide](#).

4. BladeBuilder

4.1. New Features and Enhancements

This section lists features and enhancements that are new in Release 2024 R2 of Ansys BladeBuilder.

Release 2024 R2 of BladeBuilder has no new features or enhancements.

Polyflow Release Notes

The following sections contain release information for Ansys Polyflow 2024 R2.

1. [New Features](#)
2. [Supported Platforms](#)
3. [Resolved Issues and Limitations in Ansys Polyflow 2024 R2](#)

1. New Features

The new features in Ansys Polyflow 2024 R2 are as follows:

- **MUMPS Linear Solver**

Using the MPI version of the MUMPS linear solver, calculations can now be performed in distributed memory.

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 2024 R2

This section lists limitations that existed in previous releases, but that were removed in Ansys Polyflow 2024 R2.

Forte Release Notes

The enhancements and defect corrections listed below are for Release 2024 R2.

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

1. New Features and Enhancements

This section lists new features and enhancements in Release 2024 R2 of Ansys Forte CFD, organized by topic.

Simulation Interface

- Exposed a new option to automatically calculate the surface proximity for each gap feature defined under the **Mesh Controls** node (955528).
- Exposed a new **Block Flow Across Gap** option for each gap feature control under Mesh Controls for mitigating flow leakage through the gaps (959238).
- For the Unburned Calculation Method on the Flame Speed Model panel, which is used by the G-equation flame propagation model, set the **Volume Search with Variable Radius** option as the default (967792).
- Added a checkbox under **Output Controls > Spatially Averaged And Spray** to allow spatially averaged solutions to be saved for selected parameters (1000466).

Engineering Models and Computation

- Implemented a flow blockage model at gaps to mitigate flow leakage through the gaps. It is recommended to turn on this model for gaps associated with FSI (spring-loaded valves or reed valves). For non-FSI gaps, this model is turned off by default (959238).
- Added a friction energy term in the porous media source model. This term adds thermal energy to the flow going through the porous region as an energy source, which accompanies the momentum sink (1010430).

Visualization and Analysis

- Allow the statistical quantities (mean and standard deviation) of selected spatially-averaged parameters to be saved in `thermo_stat.csv`. The parameters include temperature, pressure, equivalence ratio, and residual fraction (1000466).

Tutorials and Verification Manual

- A new rotary compressor tutorial is added, featuring the flow blockage across a reed valve which motion is computed via the built-in FSI.

Documentation

- Documented in the Forte User's Guide the fluid temperature calculation at the inflow boundaries (872341), the build-in FSI spatially averaged outputs (953673), system coupling usage (956066, 961125).
- Renewed and integrated the Forte Best Practices and Forte User's Guide manuals content. New troubleshooting sections with detailed examples and steps on how to identify and diagnose recurrent issues. Focus on engine simulations (994520) and system coupling analyses (957373), and new Best Practice added for generic compressor simulation (1042653) and rotary-lobe compressors (1042656). Additional guidance on operating environment recommendations, how to handle surface meshes and avoid common mistakes, and post-processing using Ansys EnSight.
- New python submission scripts are provided for all the System Coupling tutorials to reflect the latest system coupling best practices (956087).

2. Resolved Issues and Limitations

For a list of issues that have been resolved in Release 2024 R2, refer to the [Resolved Issues and Limitations](#) document on the Ansys Help site.

Chemkin / Energico / Reaction Workbench / MFL

Release Notes

The following sections contain release information for Release 2024 R2 of Chemkin products:

[1. New Features and Enhancements](#)

[2. Resolved Issues and Limitations](#)

1. New Features and Enhancements

Chemkin

- New functionality to use Real Gas Property (RGP) table with Batch reactor and Plug Flow reactor models.
- Added user-defined collision kernel in the sectional method for Batch reactor, Plug Flow reactor, Premixed flame reactors, and Opposed Flow reactors.
- Added dark mode option for User Interface display.
- Added option to estimate Exhaust Gas Residual (EGR) composition in Premixed flame models.
- Added option to estimate liquid density in Multiphase models.

Reaction Workbench

- New utility for converting thermodynamic data from standard format to NASA9 format.
- New utility for converting duplicate PLOG reaction rates to single aggregate reaction.
- Added option to retain nitrogen species when extracting submechanism by removing nitrogen-containing species.
- Added option to use the InChI species identifier in addition to SMILES during mechanism merge.

Model Fuel Library

- All MFL mechanisms have been updated. Improvements include:
 - Added gas-Phase autoignition chemistry of ethylene carbonate and dimethyl carbonate, which introduces options for modeling lithium-ion battery thermal runaway.
 - Improved NO_x and soot surface chemistry.
- New pre-reduced MFL mechanisms for conventional and carbon-neutral fuels that incorporate the changes to the full mechanisms:

- Mechanisms for hydrogen, ammonia, natural gas, propane, methanol, ethanol, gasoline, diesel, jet fuel, and sustainable aviation fuel.

2. Resolved Issues and Limitations

For a list of issues that have been resolved in Release 2024 R2, refer to the [Resolved Issues and Limitations](#) document on the Ansys Help site.

FENSAP-ICE Release Notes

The following sections contain release information for Ansys FENSAP-ICE 2024 R2.

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

1. New Features and Enhancements in Ansys FENSAP-ICE

- **Impact ice density model**

The impact ice density model which estimates the ice density based on local droplet impact angle and freezing fraction is upgraded to release, introducing three parameters that can be used to tune the model. These parameters are the maximum and minimum rime ice densities, and the blending function coefficient that controls the transition between the two based on the impact angle. A special handling of runback ice that forms past impingement limits where the impact angle is 0 is introduced.

- **Unsteady solver extensions for DROP3D**

The unsteady solver for ice crystals and vapor is now enabled. The two-way mass coupling between ice crystals, droplets, and water vapor can be computed. Unsteady boundary conditions provided by a timebc.dat file for droplets and crystals are now applied as expected, to simulate variations in LWC, ICC, LWC, speed, and temperature.

- **Icing with adiabatic wall boundary conditions when using particle size distributions**

Use of adiabatic walls requires running EID and vapor simulations before ICE3D. Adiabatic wall option was limited to monodispersed droplet multishot icing simulations in the earlier versions, due to the unavailability of the combined vapor solution file with particle size distribution runs. This limitation is lifted by providing combined vapor solution data alongside combined droplet and crystal solution fields. To facilitate the combination of different bins with vapor, the vapor fields are now saved in droplet and crystal files and combined with the rest of the data in these files. As part of this update, vapor solution files will no longer be visible in the FENSAP-ICE project view. ICE3D will use vapor fields directly from droplet or crystal solution files. Vapor files will only be written out if running vapor only. This update enables using adiabatic walls when running with particle size distributions, since adiabatic walls require EID and EID requires vapor solution.

- **DROP3D convergence acceleration for meshes with very high aspect ratio cells**

Structured grids, especially those found in turbomachinery components can have very thin and long grid cells which can lead to extremely small local time steps. With such grids, the droplet equations may require too many iterations to converge properly. A new minimum local time step limit is introduced with a default value of 1 microsecond (1e-6s) to avoid getting the solution stuck in such low aspect ratio parts of the mesh. This update resolves issues like unexpected melt fraction for ice crystals in cold engine parts that does not wash away in a reasonable number of solver iterations.

- **Screen boundaries added to FENSAP-TURBO**

Screen boundaries are now accessible in TURBO run types. FENSAP-TURBO and DROP3D-TURBO will be able to apply wire mesh screen settings and compute resulting pressure and LWC drops across these boundary conditions in the grid rows.

2. Resolved Issues and Limitations

For a list of issues that have been resolved in Release 2024 R2, refer to the [Resolved Issues and Limitations](#) document on the [Ansys Help website](#).

EnSight Release Notes

The following sections contain release information for Ansys EnSight 2024 R2.

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

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

Welcome to EnSight 2024 R2!

Help System

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

1. New Features and Enhancements in Ansys EnSight

The following list highlights the enhancements and changes to EnSight in 2024 R2. In addition, we have several major enhancements that have reached Beta state and can be utilized under the guidelines of a beta feature.

- **Native Rocky Reader**

EnSight now has a direct reader for the native Ansys Rocky file format. EnSight can read both the geometry, Particles and SPH Particles, starting from the *.rocky file. Grouping, naming of parts and variables attempts to follow the structure of Rocky as much as possible. To utilize shaped particles, the Rocky solver version must be 2024 R2 or greater. In coupled solver situations, you can utilize EnSight to visualize the multiple solution in a single view.

For more details, see, [Rocky Reader in the Ansys EnSight User Manual](#).

- **Shaped Particle Support**

Previously, discrete element solutions could only be rendered as sphere shapes (scaled, but only that shape). In 2024 R2, EnSight can now handle shapes to be used when rendering discrete element solutions. In the [Node Representation dialog](#), you can utilize given shapes, with orientation, scale,

and location. This works natively for the Rocky reader, but can also be utilized through other readers (for example, EnSight Case Gold).

- **Lighting Improvements**

More lighting and visual aspects of EnSight have been updated. [Shadows](#) are now shown in the OpenGL rendering. The Quad (Area) light now has handles in the viewport for interactive placement. Some [ground plane](#) defaults in interactive control interface have been changed. The control of the [environment map](#) rotation user interaction/control. has been improved. In addition, the environment map rotation is now relative to the model space (rather than screen space).

- **DVS Improvements**

Development and expansion continues for the DVS data transfer mechanism. Enhancements in 2024 R2 include single writer/multiple reader cache. Memory usage and read performance for datasets with same coordinates/connectivity/variables has been improved. Start and read connections when running in in-situ modes has been improved. Handling of dynamic part lists in DVS to EnSight has been improved with a reinit call sequence.

- **Qt Library Update**

EnSight continues to update both Qt and PyQt Libraries to improve security vulnerabilities and maintain consistency across other Ansys applications.

- **VTK Library Update**

Update to the latest VTK Library to handle newer flavors of the file format.

- **Parent and Dependent Children in Calculator Operations**

A change in the [calculator](#) logic has been introduced, when the provided parts for a calculator function include parts containing parents and their dependent children. There was a potential ambiguity as to whether that calculation should be done explicitly in the parts, or whether the dependent children should inherit their values (consistent with other operations). In 2024 R2, the dependant children are automatically removed when their parents are part of the selection for the calculator function. This means the values on the dependent children will inherit the variable value from their parent.

- **Autodyn Reader Improvements**

We have updated the Autodyn Legacy Reader to be able to handle recent Autodyn datasets.

Note:

The reader labeled as **Autodyn reader** did not receive the updates. Use the Autodyn Legacy Reader.

2. PyEnSight

The following list highlights the enhancements and changes to PyEnSight in 2024 R2.

- **Utils Module**

Enhanced utility module containing Force Tool operation as well as Pythonic calculator syntax. This allows you to leverage EnSight's calculator in a native Python syntax.

- **Omniverse Integration**

Utilization of EnSight Omniverse is now available via PyEnSight. This allows the viewport of EnSight to be sent to Omniverse and leverage its rendering, asset integration, visual material and so on. for photorealistic rendering and AR/VR display capabilities.

- **VS Code Extension**

The VS Code Extension for PyEnSight is now publicly available. This allows you to utilize the VS Code IDE for debugging and running via PyEnSight. This allows in-window display as well as auto complete, in-line help, and leverage all the debug capabilities in VS Code.

3. Ansys Dynamic Reporting (Nexus)

The following list highlights the enhancements and changes to Ansys Dynamic Reporting in 2024 R2.

- **New Plot Types**

- Heatmap
- Sankey
- Parallel Coordinates

- **Standalone Installer**

Install just the ADR components straight from the Ansys installer package.

- **Discovery Geometry**

View geometry from dsc0 files. Variables are not supported.

- **Command Line Tools**

The nexus and nexus_launcher command line tools have been renamed: adr and adr_launcher. The previous names have been retained for backward compatibility.

- **PowerPoint Export Enhancements**

The following enhancements were made in the PPTX exporter:

- Hierarchical relationship and custom slide title support for PowerPoint exports.

4. Beta in This Release

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

Default Visualization

New defaults for the visualization window are being trialed. This includes things like a new background color, new surface property defaults, ground plane, lights, and material texture applied by default. The goal is improve the default visualization and not require you to adjust multiple items (which have been available for a while) to achieve a better initial visualization.

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 2024 R2, refer to the [Resolved Issues and Limitations](#) document on the [Ansys Help website](#).

Ansys Electronics Products

Release notes are available for the following Ansys Electronics products:

[Icepak \(p. 141\)](#)

Icepak Release Notes

Release 2024 R2 of the Ansys Icepak application offers most of the capabilities from previous releases plus an enhancement.

- [Introduction \(p. 141\)](#)
- [New and Modified Features in Ansys Icepak 2024 R2 \(p. 141\)](#)
- [Resolved Issues and Limitations in Ansys Icepak 2024 R2 \(p. 141\)](#)

1. Introduction

Ansys Icepak 2024 R2 is a release of Ansys Icepak that has a new feature and resolved issues and limitations.

2. New and Modified Features in Ansys Icepak 2024 R2

Release 2024 R2 of Icepak has no new features or enhancements.

Ansys 24.2 will be the final release of the Classic Icepak platform. Classic Icepak will continue to be accessible for download beyond the 24.2 release from the Ansys Customer Portal. ACE support will formally continue for Classic Icepak until 31 December 2025.

3. Resolved Issues and Limitations in Ansys Icepak 2024 R2

For a list of issues that have been resolved in Release 2024 R2, refer to the [Resolved Issues and Limitations](#) document on the [Ansys Help website](#).

Ansys Optical Products

Release notes are available for the following Ansys optical products:

[Ansys SPEOS \(p. 145\)](#)

[SPEOS for NX \(p. 153\)](#)

[SPEOS for Creo Parametric \(p. 159\)](#)

[Ansys Optical Labs \(p. 163\)](#)

[SPEOS HPC \(p. 165\)](#)

Speos Release Notes

The following sections present the new features, enhancements and changes for Speos 2024 R2 release.

- 1. [New Features](#)
- 2. [Enhancements](#)
- 3. [Changes](#)
- 4. [Resolved Issues and Limitations](#)

1. New Features

This section describes the new features introduced in Speos 2024 R2 release.

Features from Beta to Official

- **Simulation:** The [Propagation error analysis](#) option is now officially available and no longer in BETA mode.

The Propagation error analysis option helps you identify the root causes and decrease the percentage of errors in simulation. A *.lpf file dedicated to store only the rays in error is generated out of the simulation and analyzed. Through this file you can display the rays in error in the 3D view and the type of error so that you can correct your optical system.

- **Optical Surface:** The [Photometry tool](#) is now officially released and no longer in BETA mode in the Feature Viewer of the Optical Surface.

The Photometry tool generates a photometric preview displayed in the feature viewer, a HTML report and a XMP file of the selected element(s) of the feature in order to help you verify if your current Optical Surface design passes the regulation standards defined.

- **Optical Lens:** The [Photometry tool](#) is now officially released and no longer in BETA mode in the Feature Viewer of the Optical Lens.

The Photometry tool generates a photometric preview displayed in the feature viewer, a HTML report and a XMP file of the selected element(s) of the feature in order to help you verify if your current Optical Lens design passes the regulation standards defined.

- In [Workbench](#): creating a connection between the Solution cell of a Static Structural system and the Geometry cell of a Speos system is now officially release and no longer in BETA mode.

UX

- Speos has been improved to provide a better user experience when launching a simulation: now you can **quickly identify errors** without waiting for the initialization to mesh the geometries.

When launching a simulation, the initialization does the simulation check before meshing the geometries. The initialization is stopped if system errors (optical properties not set, inconsistent set of parameters, etc.) are encountered. Errors are then displayed (to the end-user).

GPU Compatibility

- **Camera Sensor:** Simulations using a [Camera Sensor](#) based on a ***.OPTDistortion Binary file** are now compatible with GPU Compute.
- **Optical Design Exchange:** Simulations using [Optical Design Exchange](#) components or Light Box based on Optical Design Exchange components are now compatible with GPU compute.

Source

- **Surface Source:** You can now [simulate the LED flickering effect of Surface sources](#) in Inverse simulations by defining the variation of the source's power in time, with two new Surface source Timeline parameters.

Sensor

- The **Optical Library** now provides you with a [FLIR Thermal Camera package](#) containing five complete sets of FLIR Thermal Camera, a Thermal Camera Wrapper, and a Thermal Camera Editor. Thanks to this full new workflow you can generate thermal maps out of Speos Irradiance maps, and analyze them using Virtual Photometric Lab.

Component

- **Optical Design Exchange:** The [Optical Design Exchange](#) feature now supports the following types of surfaces to define lenses after exporting a design from Ansys Zemax OpticsStudio.
 - Extended polynomial
 - Zernike Standard Sag
 - Zernike Fringe Sag
 - Biconic
 - Biconic Zernike
- **Optical Design Exchange:** The [Optical Design Exchange](#) feature has been improved on many different aspects to provide a better user experience:
 - The default **meshing parameters** of the feature have been adjusted for a better precision. Parameter values are now:
 - Fixed sag of 0.001mm
 - Fixed step of 1mm
 - Meshing angle of 12 deg

- The **User Interface** of the feature definition has been improved to display only parameters related to the type of geometry/object.
- The **list of coefficients** (aspheres, Q-type, polynomials, Zernike coefficients, etc.) are now displayed in the definition panel of the feature.
- **Units** are now considered in the *.odx file and displayed in the definition panel of the feature.
- **VOP** and **SOP** defined in Ansys Zemax OpticStudio (coating) are the materials properties considered in the Speos simulation and cannot be overridden by VOP/SOP defined in Speos.
- However, you can now define **surface properties with FOP materials** on geometries faces of the Optical Design Exchange feature. In this case, the default ODX material will be overridden by the FOP that has been defined on one or more faces of the geometry.
- Simulations using Optical Design Exchange components or Light Box based on Optical Design Exchange components are now compatible with **GPU compute**.
- When importing an Optical Design Exchange feature, the position and orientation of the lense are better supported.
- The **ComponentOpticsStudio** API function allows you to access the name and parameters of each **OpticStudioGeometry**.

Stray Light Analysis

- The **Sequence Detection tool** has been fully redesigned to propose more relevant information and a user-friendly interface.
 - The list of sequences now contains new columns such as **Peak, Average, Spec R, Lamb R, Gauss R, Spec T, Lamb T, Gauss T**.
 - When clicking a sequence, the list of interactions now displays only the interactions of the sequence selected.
 - When defining an **Irradiance sensor**, you can now sort the sequences per decreasing **Energy** or per decreasing **Peak** to identify bright spots more easily.
 - The **Sequence Detection Filter** can now filter the sequences through the use of thresholds.

Live Preview

- The Live Preview has been improved to provide you with more capabilities and a better user experience:
 - A **Status Bar** now displays different information regarding the simulation that can be useful to understand how the simulation is progressing and the impact of the system settings on the performance.
 - A new panel is available, the **Light Panel** that allows you to manage the light sources included in the simulation such as changing their contributions without running a new simulation, and creating different configurations.

- The [Live Preview](#) is now compatible with the **VR Observer Sensor** and the **VR Immersive Sensor**. The Live Preview VR Sensor compatibility is in [BETA mode](#) for the current release.
- The [Live Preview](#) provides you with new **navigation modes** and **scaling options**.
- The Live Preview now allows you to [create a new sensor on the fly](#) based on the definition of the active Radiance sensor, which will be located at the current viewpoint and added to the Speos tree.

Optical Part Design

- **Light Guide:** You can now [define multiple optical axes](#) along the guide curve of a light guide so that you can smoothen the variation of the lit appearance when the observer direction changes.

2. Enhancements

This section describes the enhancements introduced in Ansys Speos 2024 R2 release.

IMPORTANT CHANGES REGARDING SPEOS LIGHT BOX

- In version prior to 2023 R1, passwords used in **Speos Light Box Import / Export** were not hidden. As soon as you open a project containing passwords, they are hidden and you cannot retrieve them.

Make sure to save your passwords in a safe place **before** opening your project in 2023 R1 or subsequent version.

- From version 2023 R1, some properties have been removed from the API Automation Interface which may have an impact on your scripting projects containing Speos Light Boxes:
 - Light box export field: Password, and Active Password. Contextual Action: Generate Password.
 - Lightbox import fields: Password
 - Speos pattern fields: Password

UX

- **Light Expert:** When activating the Light Expert in the simulation definition (Direct, Inverse, Interactive), the [LXP option](#) present for each sensor in the sensors' list is automatically activated whether the sensor is already listed or added afterwards. The LXP option can still be deactivated individually.
- The [Integration Angle](#) parameter has been renamed as **Integration Angle (Direct)** and has been moved from the additional options and settings to the definition panel of the Sensors [Observer](#), [Radiance](#) and [Immersive](#).

Simulation

- **Speos tangency algorithm** has been improved to provided a better management of tangency on [multitextured materials](#) when 2 bodies are tangent, and both tangent faces have multilayer textures applied to them.

- In case of a **Direct Simulation**, the saturation effect limitation on pixels which could have led to incorrect results has been lifted.
- Default **Smart Engine** value has been set to 15 (previous value was 11) when setting the **Ray tracer precision** to **Double**.
- Better management of **Ray tracer precision**: now when the **Geometrical Distance Tolerance** is too small to be handled by the **Single** Ray tracer precision, Speos automatically switches the Ray tracer precision to **Double** to prevent from generating unexpected result with high error rate.
- Better memory estimation when multiple sources are present in a simulation in the sources list, in Light Boxes, or in Speos Patterns.
- Better support of coated files during GPU simulations when the spectrum is in the non-visible (infrared, ultraviolet).
- Improvement of the GPU simulation accuracy when the ratio between Geometrical Distance Tolerance and the distance between the origin of the system and the system itself is very small.
- Better support of GPU Inverse simulations when they use an Ambient Natural Light source.
- Better support of GPU simulations with Fast Transmission Gathering activated in a system containing Optical Polished Surface Optical Properties.

Live Preview

- Better management of the **Out Path Faces**: In the Live Preview management, the Materials are now applied before the Out Path Faces to correctly consider the Out Path Faces in the preview.
- Improvement of the Live Preview rendering when using a high resolution sensor.

Results

- The [HTML Simulation Report](#) generated after a GPU Direct or Inverse Simulation now provides GPU simulation result and data and has been aligned with the HTML Simulation Report generated with a CPU Simulation.

Optical Part Design

- **Optical Surface:**
 - The [Custom Spread](#) parameter has been removed from the Speos interface and is now always activated. X Spread and Y Spread are then now always available in the interface.

X Spread and Y Spread correspond to the ratio driving the beam spread inside the rectangle of specification along X and Y Target axes.
- **Optical Lens:**
 - The [Optical Lens](#) algorithm has been improved for faster design generation for lenses with large number of faces.

- **Light Guide:**

- The computation time has been improved when you create curved prisms using the **Start radius** and **End radius** parameters.
- **Light Guide** performance improvement: the Light Guide construction has been improved to be faster than previous releases in Speos.
- **Light Guide** performance improvement: improvement of the robustness of the Light Guide construction algorithm when a **Draft angle** is applied.

Documentation

- The documentation regarding the [Meshing](#) has been revamped to provide more relevant and useful information.

3. Changes

This section describes the operational and support changes introduced in Ansys Speos 2024 R2 release.

Geometry Modeler

- As of version 2024 R1, the **ACIS modeler is removed**.
 - The Parasolid modeler is now the only modeler available.
 - The Environment Variable to set the modeler between ACIS and Parasolid is no longer used for Speos. However it is always active in case of a SpaceClaim session.
 - The documentation has been refactored to provide a more logical structure:
 - [Geometry Modeler](#)
 - [Lightweight/Heavyweight Import](#)
 - [CAD File Import](#)
 - [SCDOC File Import](#)

Versions Update

- Windows: Windows 10 and Windows 11
Minimum NVIDIA Driver **551.61** is required from 2024 R2.

Deprecated and End of Life Features

- Exporting a **HUD Optical Analysis** (HOA) simulation is in end of life as of version 2024 R2 and no longer possible. The HOA Export feature has been removed from the interface.

4. Resolved Issues and Limitations

For a list of issues that have been resolved in Release 2024 R2, refer to the [Resolved Issues and Limitations](#) document on the [Ansys Help website](#).

Speos for NX Release Notes

The following sections present the new features, enhancements and changes for Speos for NX 2024 R2 release.

1. New Features and Enhancements
2. Changes
3. Resolved Issues and Limitations

1. New Features and Enhancements

This section describes the new features and enhancements introduced in Speos for NX 2024 R2 release.

Features from Beta to Official

- **Simulation:** The [Propagation error analysis](#) option is now officially available and no longer in BETA mode.

The Propagation error analysis option helps you identify the root causes and decrease the percentage of errors in simulation. A *.lpf file dedicated to store only the rays in error is generated out of the simulation and analyzed. Through this file you can display the rays in error in the 3D view and the type of error so that you can correct your optical system.

UX

- New useful Speos Navigator tree commands available to improve readability:
 - Possibility to [create unlimited folders and subfolders](#) in the Speos Navigator tree to organize Speos for NX features thanks to the **Create Folder** command.



- Possibility to move Speos for NX features within their category or a targeted folder to a different position.



GPU Compatibility

- **Camera Sensor:** Simulations using a [Camera Sensor](#) based on a ***.OPTDistortion Binary file** are now compatible with **GPU Simulation**.
- The [Optimization](#) feature can now be run using **GPU Simulation** for faster optimization iterations.



Component

- New feature [Speos Pattern](#) available.

Arrays of LEDs are the new standard in lighting systems. These were usually achieved by duplicating the source multiple times with several manual operations affecting Speos Tree visibility.

The **Speos Pattern** allows you to create and orientate multiple instances of a same source file (Ray file source *.ray or Speos Lightbox *.SPEOSLightBox) into a single component, so that you do not have to create the same source multiple times in Speos for NX and overload the Speos Navigator tree and so its readability.



- **3D Texture:** now the 3D view **displays the Projection direction** (-Z axis) to help you set the 3D texture in addition to the X and Y directions.



Sensor

- The **Optical Library** now provides you with a [FLIR Thermal Camera package](#) containing five complete sets of FLIR Thermal Camera, a Thermal Camera Wrapper, and a Thermal Camera Editor. Thanks to this full new workflow you can generate thermal maps out of Speos Irradiance maps, and analyze them using Virtual Photometric Lab.
- Speos for NX now permits you to **modify the resolution** to define the corresponding sampling based on the dimensions and the wavelength of the sensor. The Resolution was read-only and calculated from the other dimension/wavelength and sampling parameters before version 2024 R2.

This improvement applies to the sensors [Irradiance](#), [Intensity](#), [Radiance](#), [Immersive](#), [Observer](#), [Camera](#), and [Human Eye](#).

Simulation

- In case of a **Direct Simulation**, the saturation effect limitation on pixels which could have led to incorrect results has been lifted.
- Default **Smart Engine** value has been set to 15 (previous value was 11) when setting the **Ray tracer precision** to **Double**.
- Better management of **Ray tracer precision**: now when the **Geometrical Distance Tolerance** is too small to be handled by the **Single** Ray tracer precision, Speos for NX automatically switches the Ray tracer precision to **Double** to prevent from generating unexpected result with high error rate.
- Better support of coated files during GPU simulations when the spectrum is in the non-visible (infrared, ultraviolet).

- Better support of GPU simulations with Fast Transmission Gathering activated in a system containing Optical Polished Surface Optical Properties.
- Better support of GPU Inverse simulations when they use an Ambient Natural Light source.

Live Preview

- The Live Preview has been improved to provide you with more capabilities and a better user experience:
 - A [Status Bar](#) now displays different information regarding the simulation that can be useful to understand how the simulation is progressing and the impact of the system settings on the performance.
 - A new panel is available, the [Light Panel](#) that allows you to manage the light sources included in the simulation such as changing their contributions without running a new simulation, and creating different configurations.
 - The [Live Preview](#) is now compatible with the **VR Observer Sensor** and the **VR Immersive Sensor**. The Live Preview VR Sensor compatibility is in [BETA mode](#) for the current release.
 - The [Live Preview](#) provides you with new **navigation modes** and **scaling options**.
- Better management of the **Out Path Faces**: In the Live Preview management, the Materials are now applied before the Out Path Faces to correctly consider the Out Path Faces in the preview.
- Improvement of the Live Preview rendering when using a high resolution sensor.

Stray Light Analysis

- The [Sequence Detection tool](#) has been fully redesigned to propose more relevant information and a user-friendly interface.
 - The list of sequences now contains new columns such as **Peak, Average, Spec R, Lamb R, Gauss R, Spec T, Lamb T, Gauss T**.
 - When clicking a sequence, the list of interactions now displays only the interactions of the sequence selected.
 - When defining an [Irradiance sensor](#), you can now sort the sequences per decreasing **Energy** or per decreasing **Peak** to identify bright spots more easily.
 - The **Sequence Detection Filter** can now filter the sequences through the use of thresholds.

Results

- The [HTML Simulation Report](#) generated after a GPU Direct or Inverse Simulation now provides GPU simulation result and data and has been aligned with the HTML Simulation Report generated with a CPU Simulation.

Optical Part Design

- Light Guide:
 - You can now [define multiple optical axes](#) along the guide curve of a light guide so that you can smoothen the variation of the lit appearance when the observer direction changes.
 - Performance improvement: the Light Guide construction has been improved to be faster than previous releases in Speos.
 - Performance improvement: the computation time has been improved when you create curved prisms using the **Start radius** and **End radius** parameters.
- **Optical Surface:** The [Custom Spread](#) parameter has been removed from the Speos interface and is now always activated. **X Spread** and **Y Spread** are then now always available in the interface.

X Spread and Y Spread correspond to the ratio driving the beam spread inside the rectangle of specification along X and Y Target axes.

Automation

- Speos for NX now provides a complete [API Automation interface](#). Thanks to the Automation interface, you can automate the execution of Speos for NX tasks.

Documentation

- The documentation regarding the [Meshing](#) has been revamped to provide more relevant and useful information.

2. Changes

This section describes the operational and support changes introduced in Ansys Speos for NX 2024 R2 release.

Versions Update

- Windows: Windows 10 and Windows 11
- Minimum NVIDIA Driver **551.61** is required from 2024 R2.

Operating System

- NX 2206 is no longer supported as of version 2024 R2.
 - NX 2212 up to NX 2212 (2212.9140) service pack is now supported from 2024 R2.
 - NX 2306 up to NX 2306 (2306.8301) service pack is now supported from 2024 R2.
 - NX 2312 is no longer in BETA and now officially released from 2024 R2.
- NX 2312 up to NX 2312 (2312.4001) service pack is now supported from 2024 R2.

- NX 2406 (2406.1300) is in BETA mode as of 2024 R2.

Deprecated and End of Life Features

- Exporting a **HUD Optical Analysis** (HOA) simulation is in end of life as of version 2024 R2 and no longer possible. The HOA Export feature has been removed from the interface.
- The **Optical Part Design** [versioning](#) is no longer supported as of version 2024 R1.

Now, when opening a file with an Optical Part Design (OPD) feature created before the current software version, the OPD feature is automatically migrated into the current software version algorithm. If you need to keep design using previous version, keep using previous Speos version.

3. Resolved Issues and Limitations

For a list of issues that have been resolved in Release 2024 R2, refer to the [Resolved Issues and Limitations](#) document on the [Ansys Help website](#).

Speos for Creo Parametric Release Notes

The following sections present the new features, enhancements and changes for Speos for Creo Parametric 2024 R2 release.

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

1. New Features and Enhancements

This section describes the new features and enhancements introduced in Speos for Creo Parametric 2024 R2 release.

General

- Better management of Speos for Creo Parametric to avoid incompatibilities with installations using Open SSL programs.

Materials

- New **Surface State Plugin** feature available:

The [Surface State Plugin](#) feature allows you to define custom models that characterize specific optical properties, for instance specific diffusion models, coatings, etc.

Those models need to be coded in a dedicated plug-in dll, *.sop file, that can be used when defining:

- Surface optical properties of parts' optical properties
- Face optical properties
- Optical properties of texture mapping feature

In addition, [Lumerical Sub-Wavelength Model](#) (LSWM) can be used with the corresponding *.json file.



Sources

- **Ray File Source:** New [Pattern](#) option available.

Arrays of LEDs are the new standard in lighting systems. These were usually achieved by duplicating the source multiple times with several manual operations affecting Speos Tree visibility.

The **Pattern** option allows you to create and orientate multiple instances of a same Ray File Source into a single feature, so that you do not have to create the same source multiple times in Speos for Creo Parametric and overload the Speos tree and so its readability.



Components

- **Speos Light Box:**

[Speos Light Box](#) is a data exchange component used to define sources and geometries.

Even though Speos Light Box is encrypted, the component offers the possibility to fully secure intellectual property by black-boxing its content:

- No detail about sources and geometries inside
- Only the **bounding box** of the complete system is displayed in the 3D view
- Ray tracing **does not expose the impacts happening inside the Speos Light Box**, without changing the optical path.

Conventional ray tracing with a Speos Light Box



Ray tracing with a Speos Light Box having Black box enabled



Sensor

- The **Optical Library** now provides you with a [FLIR Thermal Camera package](#) containing five complete sets of FLIR Thermal Camera, a Thermal Camera Wrapper, and a Thermal Camera Editor. Thanks to this full new workflow you can generate thermal maps out of Speos Irradiance maps, and analyze them using Virtual Photometric Lab.

Simulation

- In case of a **Direct Simulation**, the saturation effect limitation on pixels which could have led to incorrect results has been lifted.
- Performance improvement on the initialization of simulations containing a SOP assigned to a lot of faces.
- Default **Smart Engine** value has been set to 15 (previous value was 11) when setting the **Ray tracer precision** to **Double**.
- Better management of **Ray tracer precision**: now when the **Geometrical Distance Tolerance** is too small to be handled by the **Single** Ray tracer precision, Speos for Creo Parametric automatically

switches the Ray tracer precision to **Double** to prevent from generating unexpected result with high error rate.

Results

- The [HTML Simulation Report](#) generated after a GPU Direct or Inverse Simulation now provides GPU simulation result and data and has been aligned with the HTML Simulation Report generated with a CPU Simulation.

Speos Core Simulation Compatibility

To launch a Speos for Creo Parametric simulation using the GPU resources, you must run it through Speos Core.

- **Camera Sensor:** Simulations using a [Camera Sensor](#) based on a ***.OPTDistortion Binary file** are now compatible with **GPU Simulation**.
- Better support of coated files during GPU simulations when the spectrum is in the non-visible (infrared, ultraviolet).

Documentation

- The documentation regarding the [Meshing](#) has been revamped to provide more relevant and useful information.
- The documentation regarding the [Integration Angle](#) has been revamped to provide more relevant and useful information.

2. Changes

This section describes the operational and support changes introduced in Ansys SPEOS for Creo Parametric 2024 R2 release.

Versions Update

- Windows: Windows 10 and Windows 11

Minimum NVIDIA Driver **551.61** is required from 2024 R2.

Operating System

- Version of Creo 6 is no longer supported as of version 2024 R2.
- Version of Creo 7 from 7.0.12.0 is now supported from 2024 R2.
- Version of Creo 8 from 8.0.10.0 is now supported from 2024 R2.
- Version of Creo 9 from 9.0.8.0 is now supported from 2024 R2.
- Version of Creo 10 from 10.0.3.0 is now supported from 2024 R2.

3. Resolved Issues and Limitations

For a list of issues that have been resolved in Release 2024 R2, refer to the [Resolved Issues and Limitations](#) document on the [Ansys Help website](#).

Speos Labs Release Notes

The following sections present the new features and changes for Speos Labs 2024 R2 release.

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

1. New Features and Enhancements

This section describes the new features and enhancements introduced in Speos Labs 2024 R2 release.

Virtual Photometric Lab

- You can now [save the display options](#) of a XMP map as preset and apply them to another XMP map in one click.

Virtual Human Vision Lab

- You can now [save the display options](#) of a XMP map as preset and apply them to another XMP map in one click.

Coated Surface Editor

- *.coated file data (Reflection, Transmission, Absorption) are no longer rounded to one decimal. Now data display up to 7 significative digits.

2. Changes

This section describes the operational and support changes introduced in Ansys Optical Labs 2024 R2 release.

- Windows: Windows 10 and Windows 11

Minimum NVIDIA Driver **551.61** is required from 2024 R2.

3. Resolved Issues and Limitations

For a list of issues that have been resolved in Release 2024 R2, refer to the [Resolved Issues and Limitations](#) document on the [Ansys Help website](#).

Speos HPC Release Notes

The following sections present the changes and enhancements for Speos HPC 2024 R2 release.

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

1. New Features and Enhancements

Operating System

- Speos HPC now supports the Linux version:
 - **Red Hat 8.9**
 - **Red Hat 9.3**

GPU

- Windows: Windows 10 and Windows 11
Minimum NVIDIA Driver **551.61** is required from 2024 R2.
- Linux
Minimum NVIDIA Driver **550.54.14** is required from 2024 R2.

2. Resolved Issues and Limitations

For a list of issues that have been resolved in Release 2024 R2, refer to the [Resolved Issues and Limitations](#) document on the [Ansys Help website](#).

Ansys Acoustics Products

Release notes are available for the following Ansys Acoustics Products:

[ANSYS Sound \(p. 169\)](#)

Ansys Sound Release Notes

The following sections present the new features, enhancements and changes for Ansys Sound products in the 2024 R2 release.

1. Ansys Sound: Analysis and Specification
2. Ansys Sound: ASDforEV
3. Ansys Sound: Car Sound Simulator
4. Ansys Sound: Jury Listening Test
5. Ansys Sound: VR Sound

1. Ansys Sound: Analysis and Specification

The following sections present the new feature and enhancements for Ansys Sound: Analysis and Specification 2024 R2 release.

1.1. New Feature

Looped Samples

Functionality has been introduced to allow you to create a [looped](#) sample (without clicks) from any signal.

1.2. Enhancements

Applying the Same Display Settings to Multiple Windows

The user interface has been revised to allow you to copy and paste display settings between signal windows.

Tutorial Videos

A link to installed tutorial videos is now available from the **?** menu.

1.3. Resolved Issues and Limitations

For a list of issues that have been resolved in Release 2024 R2, refer to the [Resolved Issues and Limitations](#) document on the [Ansys Help website](#).

2. Ansys Sound: ASDforEV

The following sections present the new features and enhancements for Ansys Sound: ASDforEV 2024 R2 release.

2.1. New Feature

Autostart

You can now set ASDforEV to [Auto Start](#) mode, so that the application (and, optionally, ASDforEV CANreader) will automatically start on the next Windows startup or session logon, and can be configured in such a way that the desired .SEVproject file (and .SEVCANcfg file) is automatically loaded.

New Oscillator Types

You can now [configure oscillators](#) to use triangle, sawtooth and square waveforms in addition to pure sine tones.

2.2. Enhancements

Configuring Preprocessing Formulas

UI features have been added to allow you to insert and delete operators in [vehicle data preprocessing formulas](#).

Preprocessing Formula Testing Window

A preprocessing formula [testing window](#) is now available to allow you to check the output of a vehicle data preprocessing formula in real time for a given set of inputs.

Pitch Mapper Range

New range limits have been applied to [pitch mappers](#) to improve audio performance.

Additional AVAS Loopers

You can now add up to 6 loopers when configuring [AVAS](#) sound.

2.3. Resolved Issues and Limitations

For a list of issues that have been resolved in Release 2024 R2, refer to the [Resolved Issues and Limitations](#) document on the [Ansys Help website](#).

3. Ansys Sound: Car Sound Simulator

3.1. Resolved Issues and Limitations

For a list of issues that have been resolved in Release 2024 R2, refer to the [Resolved Issues and Limitations](#) document on the [Ansys Help website](#).

4. Ansys Sound: Jury Listening Test

The following sections present the new features and enhancements for Ansys Sound: Jury Listening Test 2024 R2 release.

4.1. New Features

Pairwise Comparison

A new test method has been added to allow [Pairwise Comparison](#) between sounds.

4.2. Resolved Issues and Limitations

For a list of issues that have been resolved in Release 2024 R2, refer to the [Resolved Issues and Limitations](#) document on the [Ansys Help website](#).

5. Ansys Sound: VR Sound

The following sections present the new features and enhancements for Ansys Sound: VR Sound 2024 R2 release.

5.1. New Features

Mono Output

A new 3D mode has been added that simulates the [mono](#) signal that would be received by a single omnidirectional microphone.

5.2. Resolved Issues and Limitations

For a list of issues that have been resolved in Release 2024 R2, refer to the [Resolved Issues and Limitations](#) document on the [Ansys Help website](#).

Ansys Autonomous Vehicle Products

Release notes are available for the following Ansys Autonomous Vehicle products:

[Ansys AVxcelerate Lighting and Sensors \(p. 175\)](#)

[Ansys AVxcelerate Sensors \(p. 177\)](#)

AVxcelerate Lighting and Sensors Release Notes

The following section presents the enhancements and changes for AVxcelerate Lighting and Sensors 2024 R2 release.

- 1. [Enhancements](#)
- 2. [Changes](#)

1. Enhancements

This section describes the enhancements introduced in AVxcelerate Lighting and Sensors 2024 R2 release.

1.1. Co-simulation with AVSimulation SCANeR™ 2023.2

AVxcelerate Lighting and Sensors now supports the 2023.2 version of AVSimulation SCANeR™ for [co-simulation between AVxcelerate Lighting and Sensors and AVSimulation SCANeR™](#).

When [installing AVxcelerate Lighting and Sensors](#), you can select the AVxcelerate SCANeR Lighting and Sensors Co-Simulation Library then set the **SCANeR Installation location** to either a 2021 SCANeR folder or a 2023 SCANeR folder to get the SCANeR configuration and resources compatible with the AVSimulation SCANeR™ version that you are using (2021.1 or 2023.2).

1.2. Co-simulation with CarMaker 12.0.1

You can now perform [Co-simulation with CarMaker](#) version 12.0.1. The AVxcelerate CarMaker Lighting and Sensors Cosimulation Library has been enriched with a sample project as well as TestRuns, roads and assets for this CarMaker 12 version.

AVxcelerate Lighting and Sensors is compatible with CarMaker versions 10.2.1, 11.0 and 12.0.1. CarMaker 9 is no longer supported.

2. Changes

This section describes the changes introduced in AVxcelerate Lighting and Sensors 2024 R2 release.

2.1. Shared Web Licensing

In the 2024 R2 release, the Ansys AVxcelerate products join other Ansys software solutions for which licensing capabilities can be obtained through Shared Web Licensing. In the Ansys Licensing Settings Utility you can enable Shared Web Licensing which offers the possibility to use:

- A local or roaming license which allows you to check out the required licenses from the cloud for offline use. With this setup, licenses are run locally for a limited duration of time, and can be checked out with respect to defined user groups and license availabilities.

- A distant license which is run on the cloud and is shared between users according to user groups and license availabilities.

For more information on Shared Web Licensing, see [Ansys Shared Web Licensing User's Guide](#)

2.2. GPU Driver

We recommend you update your NVIDIA GPU driver to a specific version, refer to [AVxcelerate Lighting and Sensors Requirements](#).

2.3. Deprecated Sensors Removed

The LiDAR and ROM radar features have been deprecated in the 2021 R1 version and the ground truth sensors feature has been deprecated from the 2024 R1 version.

LiDAR and ROM radar and the ground truth sensors (*.lidar, *.radar and *.gtsensor files) are no longer provided in the libraries.

The Simulink blocks associated to these sensors have been removed from the AVxcelerate Toolbox for Simulink.

For sensors simulations, we recommend using AVxcelerate Sensors Simulator, for more information refer to [AVxcelerate Sensors Simulator User's Guide](#).

AVxcelerate Sensors Release Notes

The following sections present the new features, enhancements and changes for AVxcelerate Sensors 2024 R2 release.

- 1. New Features
- 2. Enhancements
- 3. Changes

1. New Features

This section describes the new features introduced in AVxcelerate Sensors Simulator 2024 R2 release.

1.1. IP Protection for Radar Models

To better protect your intellectual property, you can now activate IP protection for your radar models in AVxcelerate Sensor Labs, Sensors REST API or through the Sensors Simulator API. Available in the current release as a beta feature, IP protection allows you to maintain the confidentiality of your sensor component details thanks to the following changes:

- a first layer of encryption for *.radarx files which prevents its data from being read without the use of AVxcelerate Sensors applications
- a second layer of encryption for radar models and its associated files thanks to a password protection feature in AVxcelerate Sensor Labs and the Sensors REST API. When you [activate IP Protection](#) to partially or fully protect your radar model, the component details exposed in the protected radar parameters cannot be viewed or edited in AVxcelerate Sensor Labs without first [disabling protection](#), nor is a waveform report generated during the simulation regardless of the value set for **Tx Waveform Report** in the radar simulation parameters.
- the possibility to enable IP protection for your radar through feedback control to fully protect your radar and disable the generation of the Tx Waveform Report regardless of the value set for **Tx Waveform Report** in the radar simulation parameters. If you use this method, protection can no longer be removed while the simulation is being executed.

1.2. Adaptive Grid Sampling in Radar Simulation

In the current release of AVxcelerate Sensors Simulator, new radar simulation parameters must be used to define the ray spacing used during the simulation. The **Grid Sampling** parameter now replaces the **Ray Density** parameter and allows you to define:

- the **Global Grid Sampling** which assigns a **Ray Spacing** value to be applied generally to the scene.

- the **Adaptive Grid Sampling** which specifies the **Ray Spacing** to apply to objects according to the tags applied during data preparation.

With **Adaptive Grid Sampling** you can simultaneously optimize simulation performance and the accuracy of the radar in the detection of small or distant objects during simulation. By applying specific ray spacing values to certain objects, which are less than the global ray spacing value used for the rest of the scene, you can ensure that each tagged object will be hit by at least one ray during the simulation while optimizing the use of resources and consequently simulation speed.

For more information on the **Grid Sampling** parameters, including **Adaptive Grid Sampling**, see [Radar Simulation Parameters](#) and [Setting Simulation Parameters for a Radar](#).

2. Enhancements

This section describes the enhancements introduced in AVxcelerate Sensors Simulator 2024 R2 release.

2.1. Co-simulation with AVSimulation SCANeR™ 2021.1

The installers now give you the possibility to install AVxcelerate SCANeR Sensors CoSimulation, which provides all the required elements to perform sensors simulation with AVSimulation SCANeR™ 2021.1:

- the cosimulation bridge (VRXPERIENCE_SENSORS.exe),
- a SCANeR configuration,
- a set of resources with scenarios, terrains, vehicles, pedestrians and 3D objects.

For more details, refer to the following sections in the User's Guide: [Data Preparation for SCANeR](#), [Configuring a Sensors Simulation with SCANeR](#) and [Running a Simulation in SCANeR](#).

2.2. Co-simulation with CarMaker 12

AVxcelerate Sensors Simulator is now also compatible with CarMaker version 12 and CarMaker HiL version 12. CarMaker version 9 is, however, no longer supported.

The complete list of CarMaker and CarMaker HiL versions supported by AVxcelerate Sensors Simulator 2024 R2 are listed in [AVxcelerate Sensors Simulator System Requirements](#).

2.3. Improvements to LiDAR Model

The following [Material](#) surface optical properties are now taken into account in LiDAR simulations:

- the properties set in the second layer,
- when the surface type is set to **Lambertian**, the **Absorption** value.



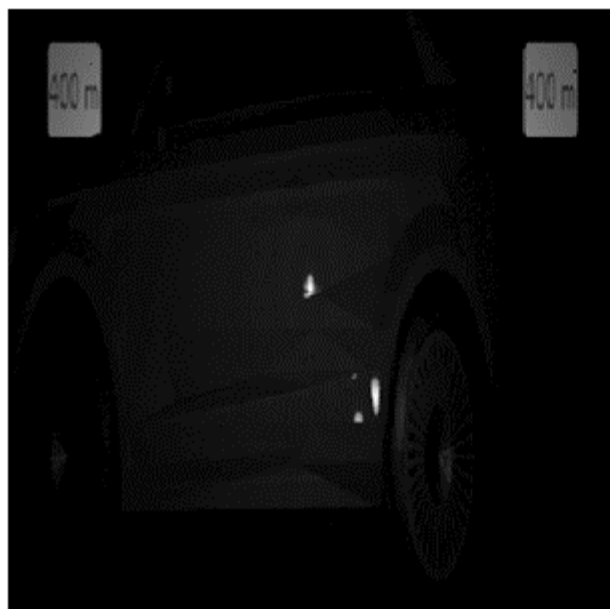
2024 R1



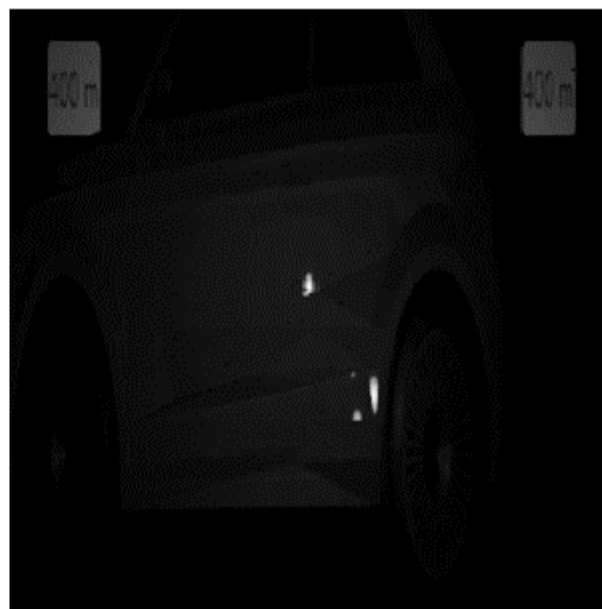
2024 R2

In addition to this enhancement, you can now configure a new [LiDAR simulation parameter](#) (**useRgbDiffuse**) so that the diffuse texture set in a **Lambertian** or **Library** layer can also be considered in LiDAR simulations.

When this simulation parameter is set to true and when the surface type is set to **Library**, the texture normalization parameter (**Color from texture** or **Color from BRDF**) defined in the surface optical properties of the material is also taken into account.



useRgbDiffuse = true
Texture Normalization = Color from texture



useRgbDiffuse = true
Texture Normalization = Color from BRDF

In this release, using the RGB diffuse texture in LiDAR simulation is delivered as a beta feature.

2.4. States in AVxcelerate Asset Preparation API

Using AVxcelerate Asset Preparation API version 3 (v3), you can now create different sets of properties, which are called states, for the following objects: materials, surface sources, point lights, directional lights, natural skies and HDRI skies.

Please refer to the [Asset Preparation API Changelog](#) section the complete list of changes introduced in AVxcelerate Asset Preparation API in the 2024 R2 release.

3. Changes

This section describes the changes introduced in AVxcelerate Sensors 2024 R2 release.

3.1. Shared Web Licensing

In the 2024 R2 release, the Ansys AVxcelerate products join other Ansys software solutions for which licensing capabilities can be obtained through Shared Web Licensing. In the Ansys Licensing Settings Utility you can enable Shared Web Licensing which offers the possibility to use:

- A local or roaming license which allows you to check out the required licenses from the cloud for offline use. With this setup, licenses are run locally for a limited duration of time, and can be checked out with respect to defined user groups and license availabilities.
- A distant license which is run on the cloud and is shared between users according to user groups and license availabilities.

For more information on Shared Web Licensing, see [Ansys Shared Web Licensing User's Guide](#)

3.2. Sensor Output Management

In AVxcelerate Sensors Simulator, for each sensor type new simulation parameters have been added to manage how the output is stored. The **Data Access Settings** parameter regroups the **Recording Format** and the new **Shared Memory Access** parameters to allow you to:

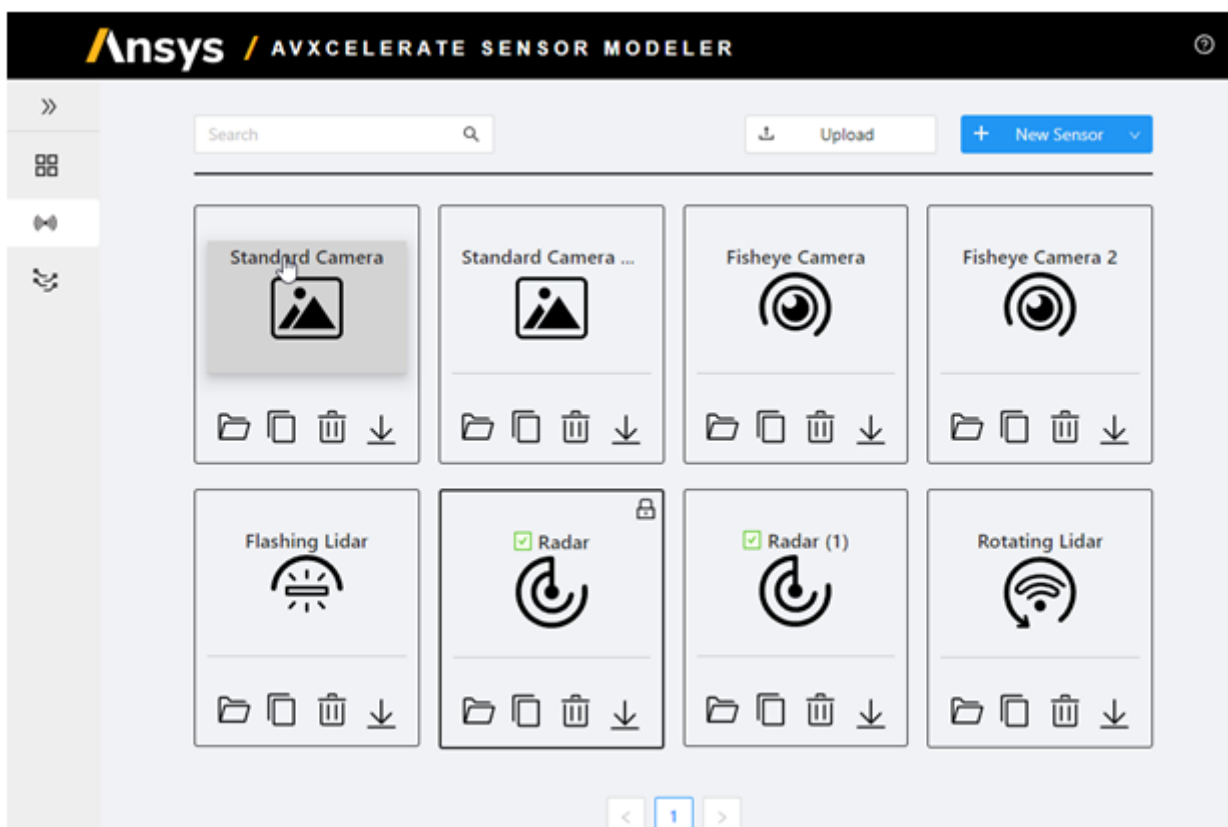
- manage whether or not sensor output is saved to the disk and its format
- disable storage to shared memory.

For more information on how the sensor results are managed, see [Results Location](#) or the relevant pages under [Simulation Parameters](#) according to the sensors you wish to simulate.

3.3. AVxcelerate Sensor Labs User Interface Changes

Several changes have been made to AVxcelerate Sensor Labs to improve the user experience.

In the Sensor Modeler you can find a new and improved main page which benefits from several changes including a redesign of buttons and sensor icons, and templates which have been regrouped into a single button for the creation of new sensors. Similar changes have also been implemented in the Sensor Layout Configurator's main page.



3.4. Power Emitted by Surface Sources

In AVxcelerate Asset Preparation Editor, the new **Flux Distribution Type** setting has been added to the [Lighting contribution parameters](#) for surface sources with the **Disjoint surfaces** option enabled.

This new parameter allows you to define how the emissive power (the **Flux** set in watt) is distributed among the different surfaces composing the surface source.

3.5. GPU Driver

We recommend you update your NVIDIA GPU driver to a specific version, refer to [AVxcelerate Sensors Simulator System Requirements](#).

3.6. VSS API Changelog

This section lists all the changes introduced in VSS API version 2024 R2 compared to version 2024 R1.

3.6.1. Changes to the Simulation service

3.6.1.1. configuration.proto

The **Configuration** message has been modified so that multiple lighting systems can now be loaded, and one of them set as active.

lighting_system is now a deprecated field, the new *LightingSystemConfiguration* field must be used instead.

The **LightingSystemConfiguration** message has been added.

In the **LightingSystem** message:

- the *identifier* field has been added,
- the bytes *lighting_system_data* pre-existing field has been moved into a oneOf,
- the *resource* field has been added to a oneOf, so that it is now possible to load a lighting system that has been uploaded using the **resourceUploader** service.

3.6.1.2. simulation_parameters.proto

The *use_rgb_diffuse* field, which is a beta parameter, has been added to the **LidarSimulation** message so that the textures of assets can be considered in LiDAR simulation.

In the **RadarSimulation** message:

- *ray_density* is now a deprecated field, the new *grid_sampling* field must be used instead,
- the new **RadarGridSampling** message has been added as well as the new **RadarGrid-SamplingParameters** message. Radar grid sampling parameters are delivered as a beta feature in the current release.

The **SensorParameters** message has been updated:

- the *recording_format* field is now deprecated, use the new *DataAccessSettings* field instead,
- the *data_access_settings* field has been added.

The **DataAccessSettings** message has been added, it includes the following two fields:

- *recording_format* (this field replaces the above-mentioned deprecated field),
- *shared_memory_access*, whose type is *boolean*. The production of sensor data in the shared memory can now be disabled.

3.6.2. New LightingSystem service

A new **LightingSystem** service has been added so that the active lighting system can be changed. This new service is exposed in the new *lighting_system.proto* file located in the *lighting_system_control* folder.

3.6.3. Changes to the Resource Uploader service

3.6.3.1. upload_data.proto

In order to expand the API's resource uploading capabilities, the **UploadMetaData** message has been updated:

- *resource_identifier* is now a deprecated field (the new below-listed field must be used instead)
- the new *values* field has been added to improve resource handling. This field enables the API to support a broader range of resource uploads, including those without a standard identifier.

3.7. Asset Preparation API Changelog

In the 2024 R2 AVxcelerate Sensors product package, the following versions of AVxcelerate Asset Preparation API are delivered: version 1 (v1), version 2 (v2) and version 3 (v3).

To ensure backward compatibility, the Asset Preparation server delivered in the 2024 R2 product package supports any of these three versions. When starting the Asset Preparation server, you must specify the API version you want to use.

3.7.1. AVxcelerate Asset Preparation API Server

This section lists the behavior-breaking changes introduced in the 2024 R2 Asset Preparation server, therefore for any version of the API (v1, v2 and v3).

- *.gif* and *.bmp* file extensions are no longer supported for textures in the **Resources** service. *images/bmp* and *images/gif* have been removed from the list of accepted MIME types.
- The identifier assigned to an object now starts at 1 (instead of 0).

3.7.2. AVxcelerate Asset Preparation API v3

This section lists all the changes introduced in AVxcelerate Asset Preparation API version 3 compared to version 2.

3.7.2.1. Changes to the Geometry service

3.7.2.1.1. geometry.proto

[Non-breaking changes] The **GetGeometryVertices** and **GetGeometryIndices** calls have been added so that you can get the vertices and the indices of a geometry.

3.7.2.1.2. geometry_description.proto

[Non-breaking changes] The following request and response messages associated with the above-mentioned new calls have been added: **GetGeometryVerticesRequest**, **GetGeometryVerticesResponse**, **GetGeometryIndicesRequest** and **GetGeometryIndicesResponse**.

[Binary breaking changes] The *name* field has been removed from the **GeometryProperties** and **MaterialPartProperties** messages.

[Protocol breaking changes] The number of each field has changed in the **GeometryProperties** and **MaterialPartProperties** messages.

[Non-breaking changes] The *name* field has been added to the following messages: **CreateGeometryRequest**, **UpdateGeometryRequest**, **CreateMaterialPartRequest**, **UpdateMaterialPartRequest**.

[Protocol breaking changes] The number of each field has changed in the following messages: **CreateGeometryRequest**, **UpdateGeometryRequest**, **CreateMaterialPartRequest**, **UpdateMaterialPartRequest**.

[Protocol breaking changes] The *identity* of the geometry is now returned by the following messages:

- **CreateGeometryResponse** message (instead of identifier)
- **GetGeometryResponse** message (instead of identifier and name)

[Protocol breaking changes] The *identity* of the material part is now returned by the following messages:

- **CreateMaterialPartResponse** (instead of identifier)
- **GetMaterialPartResponse** (instead of identifier and name)

[Protocol breaking changes] All the remaining *id* fields have been renamed *identifier*, as well as:

- *geometry_id* renamed *geometry_identifier*
- *material_part_id* renamed *material_part_identifier*
- *surface_source_id* renamed *surface_source_identifier*
- *temperature_variation_texture_id* renamed *temperature_variation_texture_identifier*

[Protocol breaking changes] The type of all the above-listed identifier fields in the request messages has changed from *int32* to *google.protobuf.Int32Value*.

[Behavior-breaking change] When the identifier of the object (or of the parent object when applicable) is not provided in a request, an error message is returned.

3.7.2.2. Changes to the Scene Tree service

3.7.2.2.1. scenetree_description.proto

[Binary breaking changes] The message **SceneTreeProperties** has been removed.

[Binary breaking changes] The *properties* field, whose type was **SceneTreeProperties**, has been removed from the **CreateSceneTreeRequest** message, this message now includes only the *name* field.

[Binary breaking changes] The *name* field has been removed from the **NodeProperties**, **GeometryInstanceProperties**, **DirectionalLightInstanceProperties** and **PointLightInstanceProperties** messages.

[Protocol breaking changes] The number of each field has changed in the **NodeProperties**, **GeometryInstanceProperties**, **DirectionalLightInstanceProperties** and **PointLightInstanceProperties** messages.

[Non-breaking changes] The *name* field has been added to the following messages:

- **CreateSceneTreeRequest** and **UpdateSceneTreeRequest**
- **CreateNodeRequest** and **UpdateNodeRequest**
- **CreateGeometryInstanceRequest** and **UpdateGeometryInstanceRequest**
- **CreateDirectionalLightInstanceRequest** and **UpdateDirectionalLightInstanceRequest**
- **CreatePointLightInstanceRequest** and **UpdatePointLightInstanceRequest**

[Protocol breaking changes] The number of each field has changed in the above-listed messages.

[Protocol breaking changes] The *identity* of the scene tree is now returned by the following messages:

- **CreateSceneTreeResponse** message (instead of identifier)
- **GetSceneTreeResponse** message (instead of identifier and name)

[Protocol breaking changes] The *identity* of the node is now returned by the following messages:

- **CreateNodeResponse** message (instead of identifier)
- **GetNodeResponse** message (instead of identifier and name)

[Protocol breaking changes] The *instance_identity* is now returned by the following messages: **CreateGeometryInstanceResponse**, **CreateDirectionalLightInstanceResponse**, **CreatePointLightInstanceResponse** (instead of identifier)

[Protocol breaking changes] The *identity* of the instance is now returned by the following messages: **GetGeometryInstanceResponse**, **GetDirectionalLightInstanceResponse**, **GetPointLightInstanceResponse** (instead of identifier and name)

[Protocol breaking changes] All the remaining *id* fields have been renamed *identifier*, as well as:

- *scene_tree_id* renamed *scene_tree_identifier*
- *node_id* renamed *node_identifier*

[Protocol breaking changes] The type of all the above-listed identifier fields in the request messages has changed from *int32* to *google.protobuf.Int32Value*.

[Behavior-breaking change] When the identifier of the object (or of the parent object when applicable) is not provided in a request, an error message is returned.

3.7.2.3. Changes to the Environment service

3.7.2.3.1. environment_description.proto

[Protocol breaking changes] in the **EnvironmentProperties** message:

- the type for the *added_skies* and *removed_skies* fields has changed, it is now *SkyLists*,
- the *active_sky* field is now a oneOf: *no_active_sky*, *active_natural_sky* or *hdri_natural_sky*.

[Non-breaking change] The **SkyLists** message has been added.

[Binary breaking changes] The *name* field has been removed from the **EnvironmentProperties** message.

[Protocol breaking changes] The number of each field has changed in the **EnvironmentProperties** message.

[Non-breaking changes] The *name* field has been added to the **CreateEnvironmentRequest** and **UpdateEnvironmentRequest** messages.

[Protocol breaking changes] The number of each field has changed in the **CreateEnvironmentRequest** and **UpdateEnvironmentRequest** messages.

[Protocol breaking changes] The *identity* of the environment is now returned by the **CreateEnvironmentResponse** message (instead of identifier) and the **GetEnvironmentResponse** message (instead of identifier and name).

[Protocol breaking changes] All the remaining *id* fields have been renamed *identifier*.

[Protocol breaking changes] The type of the *identifier* field in the request messages has changed from *int32* to *google.protobuf.Int32Value*.

[Behavior-breaking change] When the identifier of the object (or of the parent object when applicable) is not provided in a request, an error message is returned.

3.7.2.4. Sky service removed - Natural Sky Preparation and HDRI Sky Preparation services added

[Protocol Breaking change] The **Sky Preparation** service has been removed.

[Non-breaking changes] To create, get, update or delete a sky, you must now use the calls and messages provided in the following new services:

- **Natural Sky Preparation** service, exposed in *natural_sky.proto* and *natural_sky_description.proto*,
- **HDRI Sky Preparation** service, exposed in *hdri_sky.proto* and *hdri_sky_description.proto*.

Those new services include calls and messages to create, get, update and delete states for natural skies and HDRI skies.

3.7.2.5. Changes to the Material Preparation service

3.7.2.5.1. material.proto

[Non-breaking changes] New grpc calls have been added so that you can create, get, update and delete states for materials: **CreateMaterialState**, **GetMaterialState**, **UpdateMaterialState** and **DeleteMaterialState**.

3.7.2.5.2. material_description.proto

[Non-breaking changes] New request and response messages have been added for the above-mentioned new calls.

- **CreateMaterialStateRequest** and **CreateMaterialStateResponse**
- **GetMaterialStateRequest** and **GetMaterialStateResponse**
- **UpdateMaterialStateRequest** and **UpdateMaterialStateResponse**
- **DeleteMaterialStateRequest** and **DeleteMaterialStateResponse**

[Binary breaking changes] The *name* field has been removed from the **MaterialProperties** message.

[Protocol breaking changes] The number of each field has changed in the **MaterialProperties** message.

[Non-breaking changes] The *name* field has been added to the **CreateMaterialRequest** and **UpdateMaterialRequest** messages.

[Protocol breaking changes] The number of each field has changed in the above-listed messages.

[Protocol breaking changes] The *identity* of the material is now returned by the following messages:

- **CreateMaterialResponse** message (instead of identifier)
- **GetMaterialResponse** message (instead of identifier and name)

[Protocol breaking changes] All the remaining *id* fields have been renamed *identifier*, as well as:

- *surface_state_id* renamed *surface_state_identifier*
- *map_id* renamed *map_identifier*
- *volume_material_id* renamed *volume_material_identifier*
- *emissivity_variation_texture_id* renamed *emissivity_variation_texture_identifier*
- *dielectric_material_id* renamed *dielectric_material_identifier*

[Protocol breaking changes] The type of all the above-listed identifier fields in the request messages has changed from *int32* to *google.protobuf.Int32Value*.

[Behavior-breaking change] When the identifier of the object (or of the parent object when applicable) is not provided in a request, an error message is returned.

3.7.2.6. Changes to the Surface Source service

3.7.2.6.1. `surface_source.proto`

[Non-breaking changes] New grpc calls have been added so that you can create, get, update and delete material states: **CreateSurfaceSourceState**, **GetSurfaceSourceState**, **UpdateSurfaceSourceState** and **DeleteSurfaceSourceState**.

3.7.2.6.2. `surface_source_description.proto`

[Non-breaking changes] New request and response messages have been added for the above-mentioned new calls.

- **CreateSurfaceSourceStateRequest** and **CreateSurfaceSourceStateResponse**
- **GetSurfaceSourceStateRequest** and **GetSurfaceSourceStateResponse**
- **UpdateSurfaceSourceStateRequest** and **UpdateSurfaceSourceStateResponse**
- **DeleteSurfaceSourceStateRequest** and **DeleteSurfaceSourceStateResponse**

[Protocol breaking change] The field *Exitance* of the **SurfaceSourceProperties** message has been renamed *Flux*.

[Protocol breaking changes] The structure of the **LightingContribution** message has changed: the *disjoint_surfaces* field is now a oneOf: *joint* or *disjoint_flux_distribution*. The type for *disjoint_flux_distribution* is the new enum **DisjointFluxDistributionType**, which defines how the flux is distributed among the disjoint surfaces that compose the surface source.

[Binary breaking changes] The *name* field has been removed from the **SurfaceSourceProperties** message.

[Protocol breaking changes] The number of each field has changed in the **SurfaceSourceProperties** message.

[Non-breaking changes] The *name* field has been added to the **createSurfaceSourceRequest** and **UpdateSurfaceSourceRequest** messages.

[Protocol breaking changes] The number of each field has changed in the above-listed messages.

[Protocol breaking changes] The *identity* of the surface source is now returned by the following messages:

- **CreateSurfaceSourceResponse** message (instead of identifier)
- **GetSurfaceSourceResponse** message (instead of identifier and name)

[Protocol breaking changes] All the remaining *id* fields have been renamed *identifier*, as well as:

- *texture_id* renamed *texture_identifier*

- *diagram_id* renamed *diagram_identifier*
- *spectrum_id* renamed *spectrum_identifier*

[Protocol breaking changes] The type of all the above-listed identifier fields in the request messages has changed from *int32* to *google.protobuf.Int32Value*.

[Behavior-breaking change] When the identifier of the object (or of the parent object when applicable) is not provided in a request, an error message is returned.

[Change] The acceptable range for the value set in the *x* or *y* field of the *GaussianDiagram* message has been modified from $[0;\infty]$ to $[0;180]$.

3.7.2.7. Changes to the Directional Light service

3.7.2.7.1. directional_light.proto

[Non-breaking changes] New grpc calls have been added so that you can create, get, update and delete directional light states: **CreateDirectionalLightState**, **GetDirectionalLightState**, **UpdateDirectionalLightState** and **DeleteDirectionalLightState**.

3.7.2.7.2. directional_light_description.proto

[Non-breaking changes] New request and response messages have been added for the above-mentioned new calls.

- **CreateDirectionalLightStateRequest** and **CreateDirectionalLightStateResponse**
- **GetDirectionalLightStateRequest** and **GetDirectionalLightStateResponse**
- **UpdateDirectionalLightStateRequest** and **UpdateDirectionalLightStateResponse**
- **DeleteDirectionalLightStateRequest** and **DeleteDirectionalLightStateResponse**

[Protocol breaking change] The field *Power* of the **Custom** message has been renamed *Exitance*.

[Binary breaking changes] The *name* field has been removed from the **DirectionalLightProperties** message.

[Protocol breaking changes] The number of each field has changed in the **DirectionalLightProperties** message.

[Non-breaking changes] The *name* field has been added to the **CreateDirectionalLightRequest** and **UpdateDirectionalLightRequest** messages.

[Protocol breaking changes] The number of each field has changed in the above-listed messages.

[Protocol breaking changes] The *identity* of the directional light is now returned by the **CreateDirectionalLightResponse** message (instead of identifier) and the **GetDirectionalLightResponse** message (instead of identifier and name).

[Protocol breaking changes] All the remaining *id* fields have been renamed *identifier* and the *spectrum_id* field has been renamed *spectrum_identifier*.

[Protocol breaking changes] The type of all the above-listed identifier fields in the request messages has changed from *int32* to *google.protobuf.Int32Value*.

[Behavior-breaking change] When the identifier of the object (or of the parent object when applicable) is not provided in a request, an error message is returned.

3.7.2.8. Changes to the Point Light service

3.7.2.8.1. *point_light.proto*

[Non-breaking changes] New grpc calls have been added so that you can create, get, update and delete point light states: **CreatePointLightState**, **GetPointLightState**, **UpdatePointLightState** and **DeletePointLightState**.

3.7.2.8.2. *point_light_description.proto*

[Non-breaking changes] New request and response messages have been added for the above-mentioned new calls.

- **CreatePointLightStateRequest** and **CreatePointLightStateResponse**
- **GetPointLightStateRequest** and **GetPointLightStateResponse**
- **UpdatePointLightStateRequest** and **UpdatePointLightStateResponse**
- **DeletePointLightStateRequest** and **DeletePointLightStateResponse**

[Binary breaking changes] The *name* field has been removed from the **PointLightProperties** message.

[Protocol breaking changes] The number of each field has changed in the **PointLightProperties** message.

[Non-breaking changes] The *name* field has been added to the **CreatePointLightRequest**, **UpdatePointLightRequest** and **CreatePointLightStateResponse** messages.

[Protocol breaking changes] The number of each field has changed in the above-listed messages.

[Protocol breaking changes] The *identity* of the point light is now returned by the following messages:

- **CreatePointLightResponse** message (instead of identifier)
- **GetPointLightResponse** message (instead of identifier and name)

[Protocol breaking changes] All the remaining *id* fields have been renamed *identifier*, and *spectrum_id* renamed *spectrum_identifier*.

[Protocol breaking changes] The type of all the above-listed identifier fields in the request messages has changed from *int32* to *google.protobuf.Int32Value*.

[Behavior-breaking change] When the identifier of the object (or of the parent object when applicable) is not provided in a request, an error message is returned.

[Change] The acceptable range for the value set in the x or y field of the *GaussianDiagram* message has been modified from $[0;\infty]$ to $[0;180]$.

3.7.2.9. Changes in the tag.proto file

[Protocol breaking changes] Two fields of the **Properties** message have been renamed:

- *id* has been renamed *identifier*
- *irradiance_map_id* has been renamed *irradiance_map_identifier*

Ansys Mesh Prep Products

Release notes are available for the following Ansys Mesh Prep products:

[Meshing \(p. 195\)](#)

[ICEM CFD \(p. 199\)](#)

Meshing Application Release Notes

Release 2024 R2 of the Meshing application contains many new features and enhancements. Areas where you will find changes and new capabilities include the following:

1. [Changes in Product Behavior from Previous Releases](#)
2. [Geometry Fidelity](#)
3. [Enhancements to Automatic \(PrimeMesh\)](#)
4. [Enhancements to Mesh Workflows](#)
5. [Enhancements to Tetrahedrons Method](#)
6. [Enhancements to Feature Detection](#)
7. [Enhancements to Mesh Quality Worksheet](#)
8. [Enhancements to Weld Control](#)
9. [Enhancements to MultiZone Method](#)
10. [Enhancements to Quad Layer](#)
11. [Enhancements to Mesh Copy](#)
12. [Enhancements to Meshing Options](#)
13. [Enhancements to Mapped Meshing](#)
14. [Enhancements to Repair Topology](#)
15. [Enhancements to Sweep Method](#)

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 **2024 R2** release includes the following changes in product behavior:

- **Mesh Quality Worksheet** is used as a post-processing tool, when **Check Mesh Quality** is set to **No**, **Yes, Errors and Warnings** or **Yes, Errors**. You need to manually recompute the error and warning limits in the **Mesh Quality Worksheet**.
- **Mesh Copy** allows you to scope bodies.
- State of objects in the **Mesh Workflows** have been modified. For more information, See [Mesh Workflow States](#).
- In welds, only **Mesh Independent Welds** works without the **Automatic (PrimeMesh) method**.
- **Repair Topology** now requires scoping for **Repair Thin Faces Options**, **Repair Short Edges Options**, **Repair Sharp Angle Options**, and **Repair Pinch Faces Options** .

2. Geometry Fidelity

This is the first release for **Geometry Fidelity**. **Geometry Fidelity** ensures the nodes of the mesh lie on the scoped geometry including midside nodes for quadratic elements. For more detailed information, see [Geometry Fidelity](#).

3. Enhancements to Automatic (PrimeMesh)

Automatic (PrimeMesh) has been enhanced to:

- Provide **Auto Repair Topology** under **Advanced** to automatically repair topologies of the scoped bodies.

For more detailed information, see [Automatic \(PrimeMesh\)](#).

4. Enhancements to Mesh Workflows

Mesh Workflows has been enhanced to:

- Provide **Mesh Settings** options to create mesh based on acoustic settings. You can right-click **Add Settings** in **Steps** to add **Mesh Settings** in the **Steps Detail** view and **Delete Settings** to delete **Mesh Settings** in the **Steps Detail** view. You can also use **Propagate Settings** to transfer **Mesh Settings** when you update some sizes or number of layers associated with individual steps in the **Mesh Workflows**.
- Create **Hemispherical Enclosure** for **BEM Acoustics**.
- Provide **Topology Connection** in **Hole Filling** operation.
- Assign material to the zones in **Assign Zone Materials** and provide **Assign Zone Thickness operation**.
- Provide **Domain Browser** to view the parts, zones, or labels involved in the mesh workflows and to edit individual controls in mesh workflows.
- Provide **Quad Based Remeshing** in **BEM Acoustics** mesh workflow.

For more detailed information, see [Mesh Workflows](#).

5. Enhancements to Tetrahedrons Method

In **Tetrahedrons Method, Patch Conforming Algorithm** has been enhanced to:

- Provide **Refine at Thin Section** under **Refinement** to refine the number of element layers in the thin sections of the geometry.
- Provide **Refine Surface Mesh** under **Refinement** to refine the surface mesh in the thin section of geometry.

For more detailed information, see [Patch Conforming Algorithm](#).

6. Enhancements to Feature Detection

Feature Detection has been enhanced to provide mesh treatment for **Solid Fillet Rib Edges** and **Shell Fillet Rib Edges**.

7. Enhancements to Mesh Quality Worksheet

Mesh Quality Worksheet has been enhanced to provide right-click options for **Quality Criterion**:

- **Open Troubleshooting Documentation.**

Mesh Quality Worksheet has been enhanced to provide icons for **Mesh Quality Options**, **Increasing the Number of Layers** and **Decreasing the Number of Layers** for the **Quality Criterion**.

For more detailed information, see [Mesh Quality Worksheet](#).

8. Enhancements to Weld Control

Weld control has been enhanced to provide:

- Solid Solid weld connection for **Mesh Independent** weld.
- **Export ...> Weld Definition Files** option from **Weld** object on the tree.
- **Layer Transition Type** to specify the mesh element type created during layer transition.

For more detailed information, see [Weld](#).

9. Enhancements to MultiZone Method

MultiZone Method has been enhanced to:

- Provide **Program Controlled** under **Decomposition Type** to automatically determine decomposition type for each body and meshes them accordingly.
- Improve **Program Controlled** under **Surface Mesh Method** to automatically use **Prime Quad Dominant Method(Beta)** for meshing surfaces with more than 1000 patches

For more detailed information, see [MultiZone Mesh Method](#).

10. Enhancements to Quad Layer

Quad Layer has been enhanced to:

- Provide **Auto-Defeature** option to partially suppress edges or edges around the holes.
- Provide **Preview > Quad Layer** to preview the quad layers before mesh generation.
- **Layer Transition Type** to specify the mesh element type created during layer transition.

For more detailed information, see [Quad Layer](#).

11. Enhancements to Mesh Copy

Mesh Copy has been enhanced to improve performance and allows body scoping.

For more detailed information, see [Mesh Copy](#).

12. Enhancements to Meshing Options

Meshing Options has been enhanced to include **Redraw Graphics after Each Mesh Generation** to display the graphics for every step in the Mesh Worksheet after completing the mesh generation for the respective step.

For more detailed information, see [Meshing Options on the Options Dialog Box](#) .

13. Enhancements to Mapped Meshing

Mapped Meshing is enhanced to provide **MultiZone Semi-Structured** to improve mesh distribution in **MultiZone** meshing. The **Transition Type** available are **Transition (Default)**, **Circle**, **Half-Circle**, **Quarter-Circle**.

For more detailed information, see [Setting Basic Face Meshing Controls for Mapped Meshing](#).

14. Enhancements to Repair Topology

Repair Topology is enhanced to provide **Feature Angle** to specify the minimum dihedral angle at which the geometry features are repaired.

For more detailed information, see [Repair Topology](#).

15. Enhancements to Sweep Method

Sweep Method is enhanced to provide **Fall-Back Mesh Type** to select the mesh type to be used when the **Axisymmetric Sweep** fails.

For more detailed information, see [Sweep Method Control](#).

ICEM CFD

Release 2024 R2 development efforts included enhancement of Ansys ICEM CFD as a standalone application as well as continued development of its underlying technology exposed within the Ansys Workbench-based Meshing application.

Ansys ICEM CFD 2024 R2 includes new features and improvements in the following areas:

1. Enhancements to CGNS Version

1. Enhancements to CGNS Version

ICEM CFD output is enhanced to support CGNS 4.3 version now.

For more information, see [CGNS](#)

Ansys Connect Products

Release notes are available for the following Ansys Connect products:

[Ansys Minerva \(p. 203\)](#)

[Distributed Compute Gateway \(p. 209\)](#)

[optiSLang \(p. 211\)](#)

Ansys Minerva

The following topics provide an overview of what is new and improved in Ansys Minerva 2024 R2:

1. New Features and Enhancements
2. Resolved Issues
3. Known Issues and Limitations
4. Documentation

1. New Features and Enhancements

New features and enhancements are available in the following areas of Ansys Minerva 2024 R2:

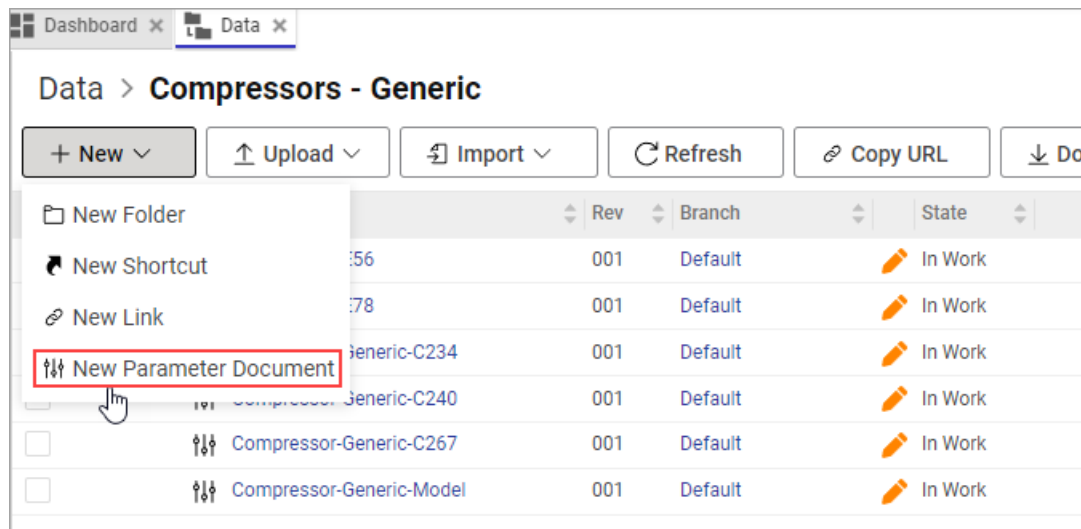
- 1.1. Data
- 1.2. Verification Plans
- 1.3. Work Requests and Tasks
- 1.4. Generic Connector

1.1. Data

- You can now create a parameter document manually and manage simulation parameters in a more efficient and standardized way without dealing with simulation files. You can use this parameter document as an input file to drive parameters in various applications, such as Ansys optiSLang, Ansys ModelCenter, and Ansys Workbench.

Furthermore, you can manage this document like a typical file in **Data** view and perform operations such as copy-paste, download, and upload on it.

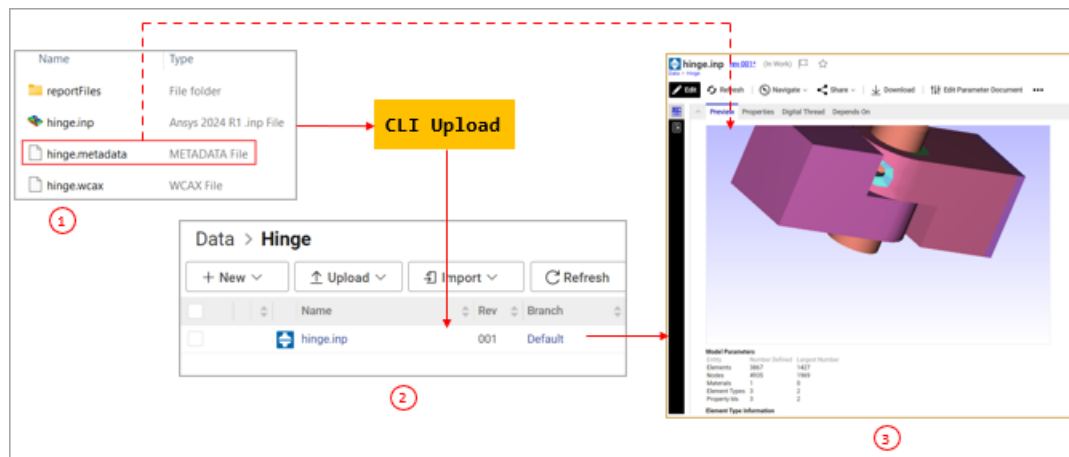
For more information, see ????.



- If you have metadata available for your simulation files locally, whether extracted through a dedicated extractor or generated by your local application, you can now upload it to Ansys Minerva with the simulation files. Ansys Minerva reads the metadata directly from the uploaded metadata files and links it with the simulation files.

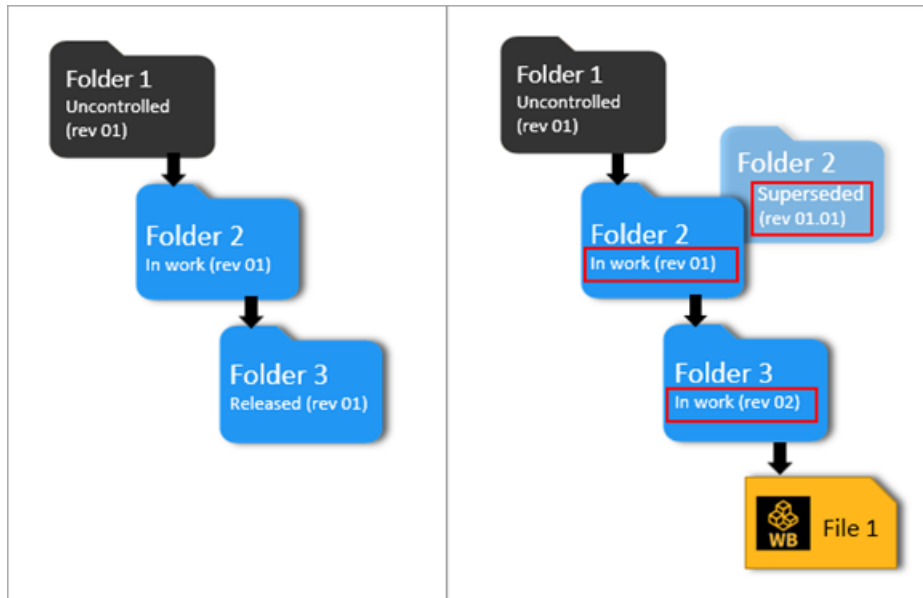
This approach enables metadata to be available quickly in Ansys Minerva as you do not have to wait for the metadata extraction service to extract the metadata.

For more information, see ????



Legends: ①: Local folder containing a .metadata file along with its associated files, ②: The Data view showing the uploaded subject file (simulation file), ③: Metadata retrieved from the uploaded .metadata file and rendered in the **Preview** tab

- When a folder structure and its revisions are crucial, such as in Ansys ModelCenter projects or Ansys STK scenarios, you can create version-controlled folders using the newly introduced **Model** classification. These folders are created directly in the **In Work** state instead of **Uncontrolled**.

Figure 3: Example: Folder Versioning

For more information, see ??? and ??? in the *Ansys Minerva User's Guide*.

Important:

Folder versioning is fully supported only with uploads using the Generic Connector.

- You can now declare a dependency as optional using the `isOptional` attribute in your dependency extraction script. For this dependency, metadata extraction will always succeed regardless of whether the dependent file is available or not. For more information, see ???.
- The dependency extraction logic for `.aedt` files has been enhanced to prevent extraction failure when corresponding folders for `$(filename).aedb` and `$(filename).aedtresults` dependencies are unavailable. For more information, see ???.

1.2. Verification Plans

The verification plans feature has been promoted from beta status to a released version. You can now seamlessly integrate verification plans into your simulation workflow to promptly validate simulation results.

Verification plans provide the following key benefits:

- Ensures accuracy and reliability of simulation results.
- Ensures that simulations meet predefined standards and criteria.
- Provides a common reference point for simulation data, criteria, and results.
- Helps identify issues early in the workflow.

For more information, see *Ansys Minerva User's Guide*: ???.

1.3. Work Requests and Tasks

- You can now convert a task to an automatic task while working in a work request. You can do this either from the **Properties** panel or from the task item view. Earlier, the creation of automatic tasks was possible only through request templates.

You can identify an automatic task by the  icon on the task node.

For more information, see *Ansys Minerva User's Guide: ????*.

- In the **Flow View** of a work request or a request template, the **Properties** panel now displays the properties of a selected task node or data transfer link when the work request or work request template is not in **Edit** mode.
- In the **Flow View** of a work request, single-user and multi-user task nodes now show the following new icons on them:
 -  or : The task is assigned to one or more users.
 - : The task requires a reviewer.
 - : The task has an assigned reviewer.

A milestone task node now displays a  icon on it to indicate that the task can be assigned to multiple users. Once the task is assigned to one or more users, the icon changes to .

1.4. Generic Connector

- The Ansys Minerva Command Line Interface (CLI) is now renamed as the Ansys Minerva Generic Connector.
- You can now use the following new commands with the Generic Connector:
 - get - local**: Fetches the working directory for a file or folder. You can then use this working directory as a value of the **--local** option in other Generic Connector commands. This helps you ensure that you work in a unified working directory and avoid creating multiple directories for files located in different folders.
 - sign-in**: Lets you sign in to Ansys Minerva and generate an access token. You can use this access token for token-based authentication, such as when executing queries or making API requests.

For more information, see *Ansys Minerva User's Guide: ????* and *????*.

- When using the **upload** command, the new **-v|--version-folders** option lets you upload folders with a specified classification. Use this option to create version-controlled folders, as the folders uploaded with this option are created in the **In Work** state instead of **Uncontrolled**.

For more information, see *Ansys Minerva User's Guide: ????*.

- The **download** command now automatically retries downloading in case of network or server errors during the download process. Furthermore, if the automatic retries are unsuccessful, you have the option to manually retry the download.

The retry mechanism makes the **download** command more efficient and robust.

- The JSON output of the Generic Connector commands has been enhanced to include details such as related items processed by the command, the absolute path to the working directory, the Ansys Minerva URL, and server connection information.

Additionally, the parameters displaying date and time in the output JSON now include a UTC offset for better comprehension.

For more information, see ????

2. Resolved Issues

For a list of issues that have been resolved in Ansys Minerva 2024 R2, see the ??? 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 ??? section of the *Ansys Known Issues and Limitations*.

4. Documentation

The *Running a Remote Desktop via EngineFrame* section has been removed from the user's guide as this information is no longer valid.

Distributed Compute Gateway (DCG)

The following topics provide an overview of what is new and improved in Distributed Compute Gateway 2024 R2:

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

1. New Features and Enhancements

No enhancements were made to Distributed Compute Gateway in this release.

2. Resolved Issues

For a list of issues that have been resolved in Distributed Compute Gateway 2024 R2, see the [????](#) 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 [????](#) section of the *Ansys Known Issues and Limitations*.

optiSLang

The following sections contain release information for optiSLang 2024 R2:

1. [Ansys Web App](#)
2. [Student Version of optiSLang](#)
3. [General](#)
4. [User Interface](#)
5. [Connectors](#)
6. [Algorithms](#)

1. Ansys Web App

The [Ansys Web App](#) has been added.

- Creates an autogenerated web app with Pre/Post and Run capabilities
- You can modify the UI using python

2. Student Version of optiSLang

optiSLang is now available as a Student Version.

- This version includes Inside Ansys, optiSLang GUI, Postprocessing and oSP3D, batch mode (command line), all APIs (pyoptiSLang, py-journal, and Plugins [Bring your own AI]).
- The Student Version is available for both Linux and Windows.
- This version can be downloaded from the [Ansys Student Versions Free Student Software Downloads](#) and the [Optimization - ANSYS Innovation Courses](#).

3. General

The following general enhancements were made:

- Performance with many Designs has been improved through several new settings.
 - Two new configuration settings: `Limit maximum designs in result design table` and `Limit number of start designs for placeholdering`
 - Project Tree widget support has been updated
 - PyOptiSLang API can now get more fine-grain result data

- Webmodule Postprocessing Module is now useable separately from optiSLang webservice
 - The module can be freely integrated in other environments
 - Currently supports optiSLang OMDB and JSON as design data sources
- OMDB file viewer was added in Minerva
 - Data from the OMDB file is situated in Minerva
 - No postprocessing server involved/needed by Minerva.

4. User Interface

The following user interface enhancements were made:

- A new "New project" wizard has been added.
 - You specify the project name and location, and the wizard creates all directories and files upfront, which greatly simplifies project setup.

5. Connectors

The following connector enhancements were made:

- optiSLang inside Fluent can now export designs and open optiSLang.
- The [Ansys EDT](#) node has new options to force SI units.
- The [Ansys HPS Node](#) node (beta) was added.
 - The HPS node allows you to log in to HPC manager, select the project, select parameters and responses, and select output files to download.
 - The [Ansys DCS](#) node is still available for older versions, though DCS itself will be retired in 2024 R2.
- The [ETK](#) node added a new separator option "Compressed".
 - Checked - separator tokens are compressed, therefore maintaining backward compatibility with old behavior
 - Unchecked - separator tokens are not compressed
- The [Thermal Desktop \(Beta\) Node](#) was added.

6. Algorithms

The following algorithm enhancements were made:

- The [Systems](#) folder has been resorted.

- The [One-Click Optimization \(OCO\)](#) has been updated:
 - Start Design settings have been added.
 - The OCO heuristic has been improved to speed up convergence.

Ansys Platform Products

Release notes are available for the following Ansys Platform products:

[ACT \(p. 217\)](#)

[Ansys Viewer \(p. 219\)](#)

[CAD Integration \(p. 221\)](#)

[Design Point Service \(DPS\) \(p. 223\)](#)

[DesignXplorer \(p. 225\)](#)

[RSM \(p. 227\)](#)

[System Coupling \(p. 229\)](#)

[Workbench \(p. 233\)](#)

ACT

No enhancements were made to ACT in 2024 R2.

Ansys Viewer

The following sections contain release information for Ansys Viewer 2024 R2:

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

1. New Features and Enhancements

No enhancements were made to Ansys Viewer in this release.

2. Resolved Issues and Limitations

See the [Resolved Issues and Limitations](#) document for information on previous limitations and key defects resolved in the current release.

3. Known Issues and Limitations

All issues and limitations known at the time of release are listed in the [Known Issues and Limitations](#) section of the *Ansys Viewer User's Guide*.

CAD Release Notes

This section summarizes the new features in CAD Integration Release 2024 R2.

For more information, see the [CAD Integration](#) section of the Ansys Help.

Rocky Linux 8.9 and 9.3 Support

The following interfaces are supported by Ansys:

- ACIS
- CATIA V5
- IGES
- Monte Carlo N-Particle
- Parasolid
- STEP

Geometry Interface Update for New CAD Releases

Geometry interfaces are updated to support new CAD releases including:

- ACIS 2024 (Reader, Windows only)
- CATIA V5 V5-6R2024 (Reader, Windows only)
- CATIA V6 2024x (Reader)
- Fusion 2.0.15995 onward
- JT 10.8 (Reader)
- NX 2312 (Reader)
- Revit 2024 (Reader)
- Rhinoceros 8 (Reader)
- Solid Edge 2024 (Reader and Plug-In)
- SOLIDWORKS 2024 (Reader)

For detailed version support information, see CAD Integration> Geometry Interface Support in the [CAD Integration](#) section of the Ansys Help.

Information about past, present and future CAD, operating system and platform support is viewable via the Ansys, Inc. website (Customer Care > Support> More Support > Platform Support).

Distributed Compute Services (DCS)

No enhancements were made to DCS in 2024 R2.

DesignXplorer

There have been no DesignXplorer changes or enhancements in this release.

Remote Solve Manager (RSM)

The following sections contain release information for Ansys Remote Solve Manager 2024 R2:

- 1. [New Features and Enhancements](#)
- 2. [Issues Resolved in this Release](#)
- 3. [Known Issues and Limitations](#)

1. New Features and Enhancements

No enhancements were made to Remote Solve Manager in this release.

2. Issues Resolved in this Release

See the [Resolved Issues and Limitations](#) document for information on previous limitations and key defects resolved in the current release.

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 [Known Issues and Limitations](#) section of the *Ansys Known Issues and Limitations*.

System Coupling

The following features have been added to the System Coupling 2024 R2 release:

1. Point Cloud for Co-Simulation
2. Electric Arc Modeling
3. Functional Mock-Up Unit (FMU) Co-Simulation Participants
4. Postprocessing
5. Graphical User Interface
6. Data Model and Commands
7. Known Issues and Limitations

1. Point Cloud for Co-Simulation

Point cloud regions for co-simulation has been moved from a hidden feature to a release feature. The following has been changed or is now supported:

- The [CouplingParticipant.Region.RegionDiscretizationType](#) setting is now available directly in the data model.
- The following coupling interfaces are supported:
 - 2D Surface Point Cloud to/from 2D Surface Point Cloud
 - 2D Surface Point Cloud to/from 2D Surface Mesh
 - 3D Surface Point Cloud to/from 3D Surface Point Cloud
 - 3D Surface Point Cloud to/from 3D Surface Mesh
 - 3D Volume Point Cloud to/from 3D Volume Point Cloud
 - 3D Volume Point Cloud to/from 3D Volume Mesh
 - FMU to Point Cloud
 - Co-simulation Parameter to Point Cloud
- Point cloud is available for:
 - Steady and transient analysis types
 - [Postprocessing](#) with EnSight
 - [Ramping](#)

- [Relaxation](#)
 - [Expressions](#) (with some limitations)
 - [Automatic alignment](#) (beta feature only with some limitations)
 - [Quasi-Newton Stabilization](#) (beta feature only)
- The SCDT Server is now able to handle transient data sets.

2. Electric Arc Modeling

System Coupling enables modeling of electric arc propagation by coupling Ansys Fluent and Ansys Electronics Desktop. System Coupling facilitates the exchange of heat losses, Lorentz forces, electrical conductivity, and temperature between these solvers in a coupled analysis workflow.

The following improvements have been made this release:

- Added support for [explicit](#) coupling participants.
- Enabled restarts for arc modeling simulations.
- Enabled geometry motion for arc modeling simulations.
- Added co-simulation parameters to C library interfaces to enable Fluent to consume parameter APIs

3. Functional Mock-Up Unit (FMU) Co-Simulation Participants

You can now add a [standard FMU participant](#) or an [FMU proxy participant](#) to the co-simulation.

4. Postprocessing

The following improvements have been made to System Coupling's postprocessing in EnSight:

- Naming of the [mapping statistics variables](#) has been streamlined.

5. Graphical User Interface

The following improvements have been made to System Coupling's graphical user interface (GUI):

- You can now select a Python script file in the Add Participant dialog box. You can also now select an input file in combination with an executable file or a Python script file.
- More options have been added to the [menu bar and toolbar buttons](#).

6. Data Model and Commands

A list of the changes made to System Coupling's data model and commands is available in the [Migrating Command-Line System Coupling Scripts](#) section of the *System Coupling User's Guide*.

7. Known Issues and Limitations

All issues and limitations known at the time of release are listed in the [Known Issues and Limitations](#) section of the *System Coupling User's Guide*.

Workbench

The Ansys Workbench platform offers many new features and enhancements. Areas where you will find changes and new capabilities include the following:

1. [General Workbench Enhancements](#)
2. [Engineering Data Workspace](#)
3. [Injection Molding Data](#)

1. General Workbench Enhancements

The following general enhancements were made for 2024 R2:

- The Ansys **Distributed Compute Services (DCS)** application has evolved into a newer family of technology components represented by [Ansys HPC Platform Services](#). The DCS application has been deprecated this release and access to it has been removed from Workbench.
- The Finite Element Modeler add-in has been unsupported from the 2019 R1 release onwards. In this release, it has been deprecated and removed from Workbench completely. Any projects that include Finite Element Modeler in the Project Schematic will not open. The Finite Element Modeler system must be removed and the project saved in releases 2024 R1 and earlier.
- The HPC Platform Services Server URL now supports both /rep and /hps addresses.
- You can now [create a connection](#) between the **Solution** cell of a Static Structural system and the **Geometry** cell of a Speos system.
- The TurboGrid system's Turbo Mesh cell has a new property, Solution Process > Update Option, which enables you to choose whether updating the cell is done in the foreground or background. For details, see [TurboGrid in the Workbench User's Guide](#).
- You can now import Mechanical solution transformation values into Workbench for use in the [Sherlock Post](#) system.

2. Engineering Data Workspace

For Release 2024 R2, the following enhancements have been made to the Engineering Data workspace:

- **Crack Growth - Paris' Law**. You can now directly choose the units of measurement for **Length per Cycle Units** and **Stress Intensity Factor Units** instead of using reference units. This allows you to enter experimental material data directly in your preferred unit system. Workbench and the solver automatically convert these values to the current unit system.

3. Injection Molding Data

For Release 2024 R2, there are no new feature updates for [Injection Molding Data](#).

Embedded Software Products

Release Notes

Release notes are available for the following Ansys embedded software products:

[SCADE Suite \(p. 237\)](#)

[SCADE Display \(p. 243\)](#)

[SCADE Test \(p. 247\)](#)

[SCADE Architect \(p. 253\)](#)

[SCADE LifeCycle \(p. 257\)](#)

[SCADE Solutions for ARINC 661 Compliant Systems \(p. 261\)](#)

[SCADE Avionics Package \(p. 265\)](#)

SCADE Suite Release Notes

The following sections present the new features and enhancements for SCADE Suite 2024 R2 release.

- 1. [New Features and Enhancements](#)
- 2. [Certification Credits](#)
- 3. [Limitations and Workarounds](#)
- 4. [Migration Information](#)
- 5. [Supported External Tool Versions](#)

1. New Features and Enhancements

This section describes the new features and enhancements introduced in SCADE Suite 2024 R2 release.

SCADE Python Extension Management

- New Extension Manager enabling discoverability, installation, and removal of Ansys SCADE Python extensions

SCADE Suite Model Design

- New **SCADE Suite Design Standard** distributed as an example for users having to develop their own design standard in their development projects
- Improved Metrics and Rules Checker with:
 - New set of design and methodological rules coming from SCADE Suite Design Standard
 - Better granularity allowing to fine-grain metric/rule application on model content
 - Capability to raise multiple errors and allow multiple justifications on a single Scade object
- Improved error messages and location of design errors

SCADE Suite UX Improvements

- Several improvements for saving design time:
 - Retrieving easily path of enumerated values
 - Keeping object references when moving a declaration from a high scope to a lower one
 - Improved management of constant values when changing type

SCADE Automotive Package

- New ISO 26262 certification credits for SCADE ACG 2.6 (see [Certification Credits \(p. 238\)](#))

SCADE Test Integration

For details, consult [SCADE Test Release Notes \(p. 247\)](#).

SCADE LifeCycle Integration

For details, consult [SCADE LifeCycle Release Notes \(p. 257\)](#).

2. Certification Credits

The SCADE Suite 2024 R2 release includes the following tool with certification credits:

- **SCADE Suite KCG 6.6.4** and **SCADE Suite KCG 6.6.2** for qualification under DO-178C/DO-330 up to TQL-1 and for certification under ISO 26262 up to ASIL D, IEC 61508 up to SIL 3, or EN 50128 up to SIL 3/4
- **SCADE ACG 2.6** for certification under ISO 26262 up to ASIL D
- See also [Certification Credits \(p. 248\)](#) for SCADE Test and [Certification Credits \(p. 257\)](#) for SCADE LifeCycle

3. Limitations and Workarounds

This section describes the limitations and workaround solutions applying to the SCADE Suite 2024 R2 release.

Limitation in Coverability Analysis

- In Model Coverability analysis report, it is not possible to switch the display of detailed results between full and compact results when the project is stored on a network disk.
- In Model Coverability analysis report, the numbering of headings does not display when the project is stored on a network disk.

Limitation in Simulink Importer

- Import does not support tunable popup parameters set using a referencing enumeration class.

Limitation in Code Generation for FMUs

- When using FMU states, any modification of model requires to save FMU states.

Limitation in Python Scripts

- Some Python extension modules (Python modules implemented in C) like numpy or lxml are not working in SCADE environment.

Limitation in SCADE Suite Modeling

- Floating windows do not operate properly when a SCADE Architect project is loaded at the same time as SCADE Suite: close and reopen SCADE Suite to resolve the issue.

Limitation in SCADE Suite Simulation

- Ada language pragmas referenced in SCADE Suite KCG “pragma” pragma may potentially not be supported for Ada code simulation.

Limitations in Code Integration

- [Ada] Provided Targets and Adaptors do not support Ada. Ada dedicated utilities can be developed using Python API.
- [C] Imported types at root level are not supported when generating wrapping code for Ansys Twin Builder.

Limitation in Multicore Code Generator

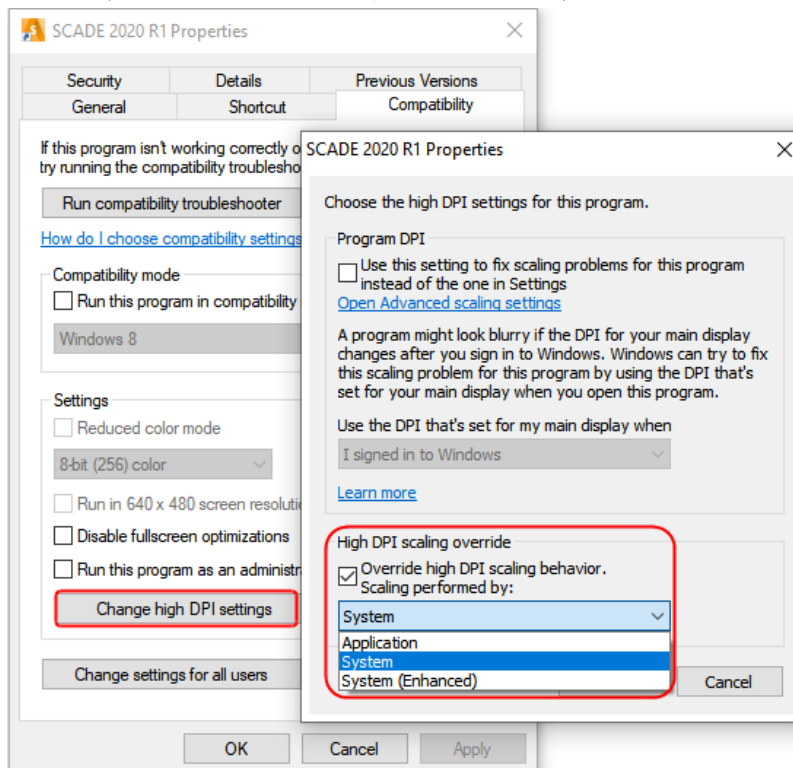
- With clocked tasks, generated code may contain cyclic dependency between methods, which prevents from integrating it.

Limitation in SCADE Applications

- SCADE applications may not scale well (with symptoms like tiny icons and UI elements) on high-DPI (high-dots-per-inch) displays, depending on the display scaling factor selected for the user's Windows display settings.

Workaround*: users can let Windows automatically manage the scaling of SCADE applications. This may result in applications being blurry, but scaled correctly.

1. Right-click the SCADE product binary (scade.exe) in File Explorer (or Desktop shortcut) to open Properties. On Compatibility tab, click **Change high DPI settings**, under High DPI scaling override section, check the **Override high DPI scaling behavior** option.
2. Select **System** for the **Scaling performed by** drop-down.



* This is only applicable to Windows 10 version “1703” (also called “Creators Update”, delivered in April 2017) and subsequent ones, and can be applied to all SCADE versions. The same operation can be applied to the vcs.exe binary in the same folder.

4. Migration Information

SCADE Suite 2024 R1 to 2024 R2

- No format change: no project migration issue
- Support for migrating Scade 6.4 projects to Scade 6.6 projects is no longer available. Loading Scade 6.4 models is no longer supported. Please contact your Ansys representative or Support if you have such need.
- In SCADE Suite Metrics and Rules Checker, the rule SCADE_R001 (for detecting non-coverable points) is replaced by a set of design and methodological rules. In case, you were using such a rule in your projects, it is recommended to launch Model Coverability analysis in SCADE Suite. If need be, the rule can be requested from support.

SCADE Suite 2023 R2 to 2024 R1

- No format change: no project migration issue

SCADE Automotive Package 2024 R1 to 2024 R2

- No format change: no project migration issue

SCADE Automotive Package 2023 R2 to 2024 R1

- When loading existing models, it is not possible to fill automatically the new accessKind attribute of the LocalDataAccess annotation. It is necessary to make the following update:
 1. Open 2023 R2 model(s) with 2024 R1.
 2. Launch AUTOSAR to Suite synchronization.
 3. Check that Diff/merge results only report differences about the value change of the accessKind attribute.
 4. Accept changes reported by Diff analysis to set the accessKind attribute from “Undefined” to “ReadLocalVariable”, “WrittenLocalVariable”, or “ParameterAccess”.
 5. Save the SCADE Suite model.

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 <scade-support@ansys.com>.

5. Supported External Tool Versions

This section describes the external tools supported by SCADE Suite 2024 R2 release.

External Tools	Version
NI VeriStand	2015 and 2017
Mathworks Simulink/Stateflow for Simulink Importer	R2011b to R2023a

SCADE Display Release Notes

The following sections present the new features and enhancements for SCADE Display 2024 R2 release.

1. [Fixes and Enhancements](#)
2. [Certification Credits](#)
3. [Limitations and Workarounds](#)
4. [Migration Information](#)
5. [Supported Graphical Libraries Version](#)

1. Fixes and Enhancements

This release of SCADE Display is a maintenance release containing fixes and enhancements.

SCADE Display Display Modeling

- New Halo option in Font Importer allowing to control font halo thickness when importing fonts
- Capability to set finer grid size when assembling HMIs: new minimum dimension of 1.0 pixel

Connection to Graphical Panels

- Graphical Panel Coupling API with full programmatic capabilities:
 - for listing graphics-side entities, logics-side entities, or existing connections by name
 - for creating/deleting connections
- Graphical Panel Coupling API supporting automated connection workflows and easy management of large embedded display projects

SCADE Test Integration

For details, consult [SCADE Test Release Notes \(p. 247\)](#).

SCADE LifeCycle Integration

For details, consult [SCADE LifeCycle Release Notes \(p. 257\)](#).

2. Certification Credits

The SCADE Display 2024 R2 release includes the following tools with certification credits.

- **SCADE Display KCG Code Generator 6.7.1:** for qualification under DO-178C/DO-330 up to TQL-1 and for certification under ISO 26262, IEC 61508 up to SIL 3, or EN 50128 up to SIL 3/4

- See also [Certification Credits \(p. 248\)](#) for SCADE Test and [Certification Credits \(p. 257\)](#) for SCADE LifeCycle

3. Limitations and Workarounds

This section describes the limitations and workaround solutions applying to the SCADE Display 2024 R2 release.

Limitations in SCADE Display Modeling

- Effect containers are incompatible with glTF objects and should not be used with them.
- No support of animations internal to glTF objects when using GltfImporter widget.
- Masks on active areas are not supported.

Limitations in Code Generation for FMUs

- When using FMU states, any modification of model requires to save new FMU states
- When using FMU states, user imported function with memory must not store pointers

Limitation in SCADE Display Simulation

- “Restart” command and “Reset” scenario command in reference object simulation: array and structure variables are not updated to the Variable Dictionary's initial value. Instead, they are initialized to 0

Limitations in SCADE Display KCG & OGLX

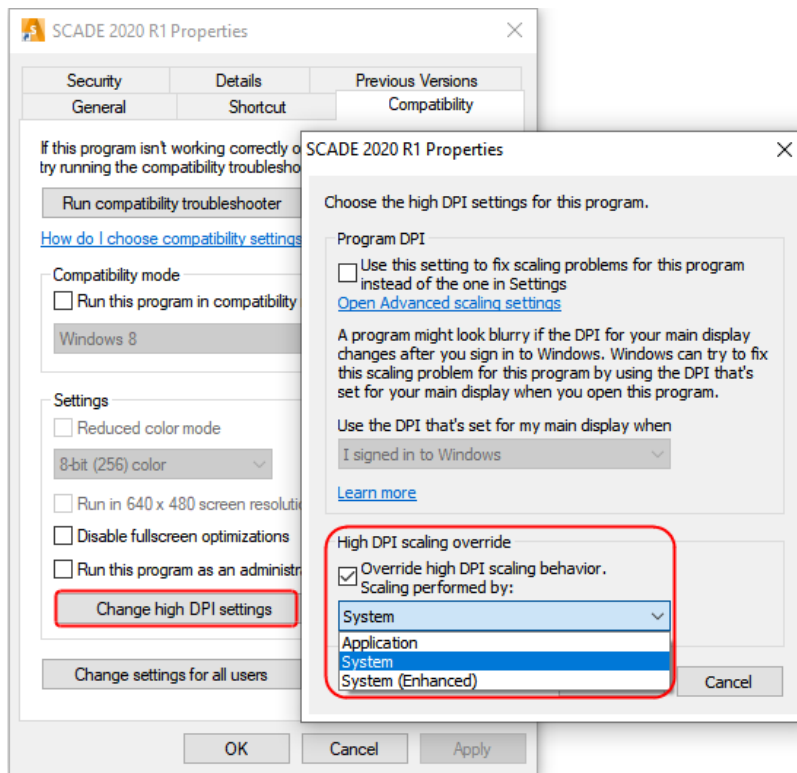
- When drawing a primitive with arc segments, and setting transparency on the outline part of the primitive, darker points appear where the elementary lines used to draw the segment are superposed.
- Clip-planes are not supported in OGLX with OpenGL driver SC 1.0.1.
- Only 16 nested containers with Push/Pop matrix supported with OpenGL driver SC 1.0.1.

Limitation in SCADE Applications

- SCADE applications may not scale well (with symptoms like tiny icons and UI elements) on high-DPI (high-dots-per-inch) displays, depending on the display scaling factor selected for the user's Windows display settings.

Workaround*: users can let Windows automatically manage the scaling of SCADE applications. This may result in applications being blurry, but scaled correctly.

1. Right-click the SCADE product binary (scade.exe) in File Explorer (or Desktop shortcut) to open Properties. On Compatibility tab, click **Change high DPI settings**, under High DPI scaling override section, check the **Override high DPI scaling behavior** option.
2. Select **System** for the **Scaling performed by** drop-down.



* This is only applicable to Windows 10 version "1703" (also called "Creators Update", delivered in April 2017) and subsequent ones, and can be applied to all SCADE versions. The same operation can be applied to the vcs.exe binary in the same folder.

4. Migration Information

SCADE Display 2024 R1 to 2024 R2

- No format change: no project migration issue

SCADE Display 2023 R2 to 2024 R1

- Format change: at loading, automatic migration of .etp project files

SCADE Display 2023 R1 to 2023 R2

- No format change: no project migration issue

SCADE Display 2022 R2 to 2023 R1

- Format change: at loading, automatic migration of .etp project files

For migration information related to previous versions, contact <scade-support@ansys.com>.

5. Supported Graphical Libraries Version

This section describes the graphical libraries supported by SCADE Display 2024 R2 release.

Graphical Library	Version
OpenGL	OpenGL 1.3 OpenGL ES 1.1 and ES 2.0 OpenGL SC 1.0.1 and SC 2.0

SCADE Test Release Notes

The following sections present the new features and enhancements for SCADE Test 2024 R2 release.

1. New Features and Enhancements
2. Certification Credits
3. Limitations and Workarounds
4. Migration Information
5. Version Compatibility for Testing Environment

1. New Features and Enhancements

This section describes the new features and enhancements introduced in SCADE Test 2024 R2 release.

Model Coverage

- New statuses about coverage points facilitating the identification of “Not Coverable” points discovered during coverability analysis or during instrumentation

SCADE Rapid Prototyper

- New Halo option in Font Importer allowing to control font halo thickness when importing fonts
- Capability to set finer grid size when assembling HMLs: new minimum dimension of 1.0 pixel
- Graphical Panel Coupling API with full programmatic capabilities:
 - for listing graphics-side entities, logics-side entities, or existing connections by name
 - for creating/deleting connections
- Graphical Panel Coupling API supporting automated connection workflows and easy management of large embedded display projects

SCADE Python Extension Management

- New Extension Manager enabling discoverability, installation, and removal of Ansys SCADE Python extensions

SCADE LifeCycle Integration

For details, consult [SCADE LifeCycle Release Notes \(p. 257\)](#).

2. Certification Credits

The SCADE Test 2024 R2 release includes the following tool with certification credits:

- **SCADE Test Environment for Host for SCADE Suite - 2024 R1** for qualification under DO-178C/DO-330 TQL-5 and for certification under ISO 26262 TCL3, IEC 61508 T2, or EN 50128 T2
- **SCADE Test Model Coverage for SCADE Suite - 2024 R1** 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 - 2024 R1** ready for qualification under DO-178C/DO-330 TQL-5 and for certification under ISO 26262 TCL3, IEC 61508 T2, or EN 50128 T2
- **SCADE Test Environment for Host for SCADE Display - 2021 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 Display - 2021 R2** for qualification under DO-178C/DO-330 TQL-5 and for certification under ISO 26262 TCL3, IEC 61508 T2, or EN 50128 T2 (Ada coverage is outside certification scope)
- **SCADE Test Target Execution for SCADE Display - 2021 R2** for qualification under DO-178C/DO-330 TQL-5 and for certification under ISO 26262 TCL3, IEC 61508 T2, or EN 50128 T2

3. Limitations and Workarounds

This section describes the limitations and workaround solutions applying to the SCADE Test 2024 R2 release.

Limitation in Coverability Analysis

- In Model Coverability analysis report, it is not possible to switch the display of detailed results between full and compact results when the project is stored on a network disk.
- In Model Coverability analysis report, the numbering of headings does not display when the project is stored on a network disk
- Internal failure occurs when analyzing a model containing an operator abstracted by mixed list of imported or non-imported operators and where several abstract operators are polymorphic in type and size.

Workaround:

1. Update the model so that an operator abstracts either imported operators or non-imported operators.
2. Update the model so that it does not contain any size polymorphic abstract operator.

Limitations in Rapid Prototyper

- No support of animations internal to glTF objects when using GltfImporter widget
- Masks on active areas are not supported

- No support for SCADE Suite KCG 6.6 Ada
- Code Generation for FMUs:
 - When using FMU states, any modification of model requires to save FMU states
 - When using FMU states, user imported function with memory must not store pointers

Limitation in Test Execution

- Snapshots not supported by test execution on SCADE Display specifications

Limitations in Test Target Execution

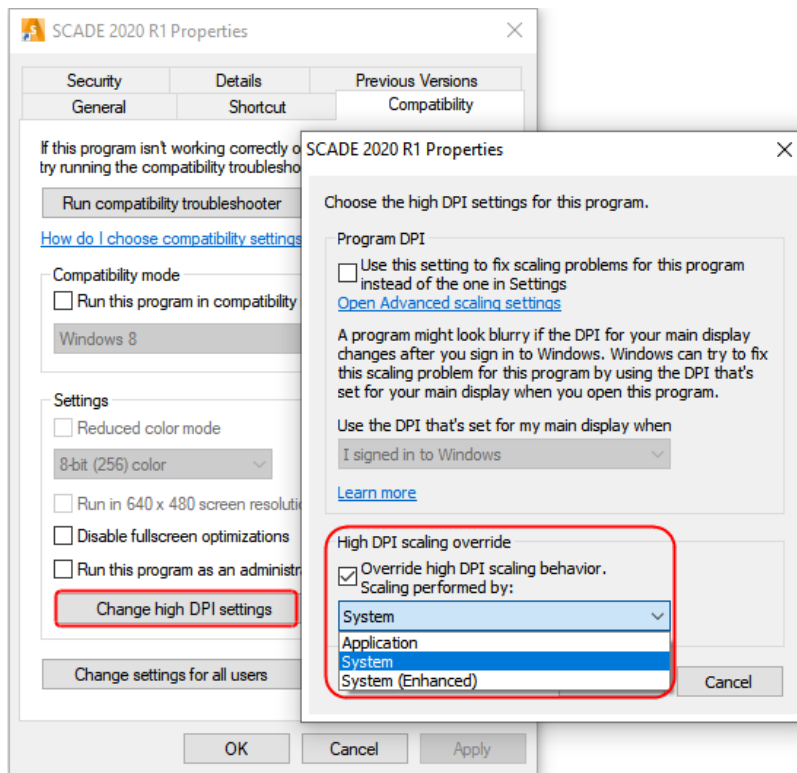
- **Test Harness Generation** (C or Ada): assignments and tests on data of external types, and snapshots are not supported
- **Test Harness for RTRT target** (C): for generated test harnesses to be properly interpreted by RTRT, `kcg_char` must be mapped to `unsigned char`
- **Test Harness for VectorCAST target:**
 - Models with sensors redefinition cannot be used by VectorCAST test harnesses (C code)
 - For generated test harnesses in C code to be properly interpreted by VectorCAST, `kcg_char` must be mapped to `char` and `kcg_true` must be mapped to 1 and `kcg_false` to 0 (default mapping)

Limitation in SCADE Applications

- SCADE applications may not scale well (with symptoms like tiny icons and UI elements) on high-DPI (high-dots-per-inch) displays, depending on the display scaling factor selected for the user's Windows display settings.

Workaround*: users can let Windows automatically manage the scaling of SCADE applications. This may result in applications being blurry, but scaled correctly.

1. Right-click the SCADE product binary (`scade.exe`) in File Explorer (or Desktop shortcut) to open Properties. On Compatibility tab, click **Change high DPI settings**, under High DPI scaling override section, check the **Override high DPI scaling behavior** option.
2. Select **System** for the **Scaling performed by** drop-down.



* This is only applicable to Windows 10 version “1703” (also called “Creators Update,” delivered in April 2017) and subsequent ones, and can be applied to all SCADE versions. The same operation can be applied to the vcs.exe binary in the same folder.

4. Migration Information

SCADE Rapid Prototyper 2024 R1 to 2024 R2

- No format change: no project migration issue

SCADE Rapid Prototyper 2023 R2 to 2024 R1

- Format change: at loading, automatic migration of .etp project files

SCADE Rapid Prototyper 2023 R1 to 2023 R2

- No format change: no project migration issue

SCADE Test 2024 R1 to 2024 R2

- No format change: no project migration issue

SCADE Test 2023 R2 to 2024 R1

- No format change: no project migration issue

SCADE Test 2023 R1 to 2023 R2

- The cmp/nocmp values are removed from the list of values of the -faq11 option and no longer accepted by SCADE Test Model Coverage for SCADE Suite

For migration information related to previous versions, contact <scade-support@ansys.com>.

5. Version Compatibility for Testing Environment

Since SCADE Test Environment relies on SCADE Suite KCG and SCADE Display KCG, the synchronization of versions is required. The following versions are compatible:

Product	Versions
Test Environment for Host	2021 R2 (Display) 2024 R1 (Suite)
SCADE Suite KCG	6.6.2 (certification credits) 6.6.4 (certification credits)
SCADE Display KCG	6.7.1 (certification credits)
SCADE Test Model Coverage	2021 R2 (Display) 2024 R1 (Suite)
SCADE Test Target Execution	2021 R2 (Display) 2024 R1 (Suite)
Target Execution support for:	
IBM Rational® Test RealTime (SCADE Display)	7.5.0.0 and 8.0.2 (C&Ada)
IBM Rational® Test RealTime (SCADE Suite)	7.5.0.0, 8.0.2, and 8.3.0.0 (C&Ada)
Vector Software VectorCAST™/ C++ (SCADE Suite only)	6.4b, 2020 SP3, and 2022 SP5 (C&Ada)
LDRA Testbed® (SCADE Suite only)	9.7.2, 9.8.3, 10.0.1 (C) or 9.4.7 (Ada)

SCADE Architect Release Notes

The following sections present the new features and enhancements for SCADE Architect 2024 R2 release.

1. [Fixes and Enhancements](#)
2. [Limitations and Workarounds](#)
3. [Migration Information](#)
4. [Supported External Tool Versions](#)

1. Fixes and Enhancements

This release of SCADE Architect is a maintenance release containing fixes.

SCADE Python Extension Management

- New Extension Manager enabling discoverability, installation, and removal of Ansys SCADE Python extensions

SCADE LifeCycle Integration

For details, consult [SCADE LifeCycle Release Notes \(p. 257\)](#).

2. Limitations and Workarounds

This section describes the limitations and workaround solutions applying to the SCADE Architect 2024 R2 release.

Limitations in SCADE Architect Modeling

- Changing type of part/reference not supported
- Turning part into reference not supported
- SCADE Architect diagrams may not display connections and associations properly (for example, alignment of wires shifting) when opening diagram with different DPI (high-dots-per-inch) displays: the option **DPI Rendering** can be used in diagram properties to correct alignment
- Definition of new Guard and Weight properties on existing Control Flows may not display in Activity diagrams: recreate the Control Flow to refresh diagram with new property information.

Limitations in SCADE Architect Application

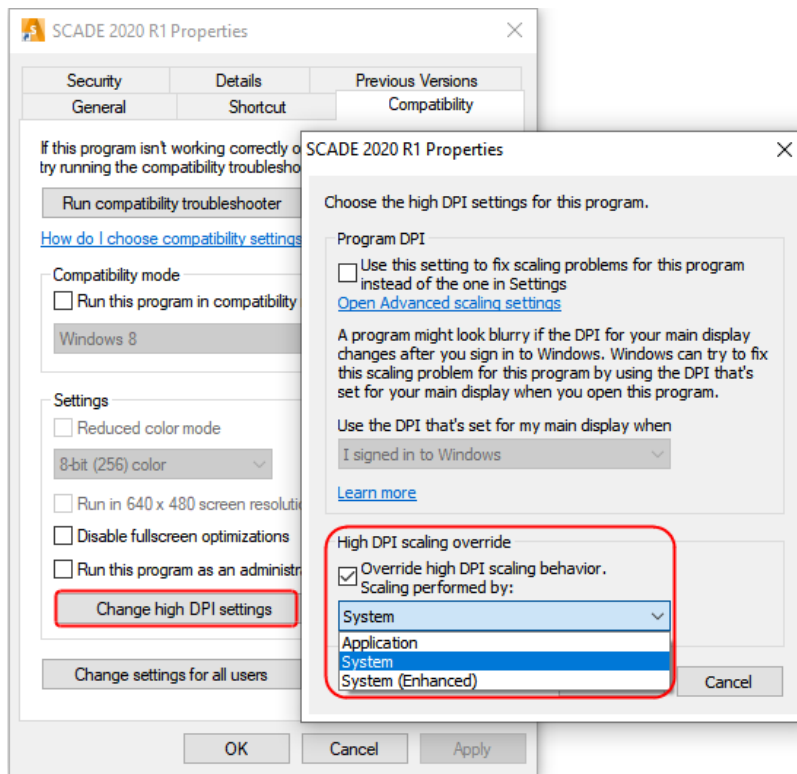
- SCADE Architect may fail to start or may allocate too little memory to its Java Virtual Machine under some rare circumstances (mostly when running in a virtualized environment with machines having less than 8 GB RAM). Default memory allocation can be fine-tuned by adding a specially crafted sc -

sys.ini file in the <INSTALLATION>/SCADE/ folder and containing the following line:
 -Xmx1024m. The value after -Xmx is given in megabytes and can be adjusted as required.

- SCADE applications may not scale well (with symptoms like tiny icons and UI elements) on high-DPI (high-dots-per-inch) displays, depending on the display scaling factor selected for the user's Windows display settings.

Workaround*: users can let Windows automatically manage the scaling of SCADE applications. This may result in applications being blurry, but scaled correctly.

1. Right-click the SCADE product binary (scade.exe) in File Explorer (or Desktop shortcut) to open Properties. On Compatibility tab, click **Change high DPI settings**, under High DPI scaling override section, check the **Override high DPI scaling behavior** option.
2. Select **System** for the **Scaling performed by** drop-down.



* This is only applicable to Windows 10 version "1703" (also called "Creators Update", delivered in April 2017) and subsequent ones, and can be applied to all SCADE versions. The same operation can be applied to the vcs.exe binary in the same folder.

Limitations in Model Import/Export

- Table query scripts, ATY files, and content providers must be explicitly added by users when exporting an archive
- Object colors in diagrams are lost when importing Enterprise Architect models

Limitations in Configurator

- Reference establishment is limited to the scope of user metaclasses. It is not possible to directly reference Architect API metaclasses.
- SCADE LifeCycle Reporter ignores configuration aspects and reports configured models as standard SCADE Architect models.
- During a working session, it is required to open only SCADE Architect projects or only SCADE Architect Configurator projects. If ever alternating projects, one must close the current session and reopen the application. A message warns you about potential conflicts or loading errors whenever switching between SCADE Architect projects and SCADE Architect Configurator projects in the same session.

Limitations in Diff/Merge

- Diagrams, Sequences, and Use Cases are only matched/ merged as a whole (no matching/merging of sub-elements).
- Notes with structured types are accessible as a whole from API and merged as a whole.

Limitation in Architect API

- SCADE Architect Java API for annotations depends on ANTLR 3.2.0 (Java component parsing textual files). The API cannot be installed on an Eclipse target platform that includes a more recent and incompatible version of the **org.antlr.runtime** plug-in. Recent versions of the XText Eclipse modeling component rely on ANTLR 4 and cannot be installed on the same platform as the API.
- SCADE Architect API has no support for concepts coming from Use Case or Sequence diagrams.

3. Migration Information

SCADE Architect 2024 R1 to 2024 R2

- No specific issue: see [SCADE Architect \(each release\)](#) (p. 255)

SCADE Architect 2023 R2 to 2024 R1

- No specific issue

SCADE Architect 2023 R1 to 2023 R2

- Very old project format used for registering Checker settings are not migrated. They may no more be recognized for rule selection in user interface.

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 <scade-support@ansys.com>.

4. Supported External Tool Versions

This section describes the external tools supported by SCADE Architect 2024 R2 release.

External Tools	Version
IBM® Rational® Rhapsody®	8.2
Sparx Systems Enterprise Architect	Up to 16.0
Dassault Systèmes® Cameo Systems Modeler™	Up to 2022xR1
Eclipse (for Architect API)	Oxygen 3a

SCADE LifeCycle Release Notes

The following sections present the new features and enhancements for SCADE LifeCycle 2024 R2 release.

1. New Features and Enhancements
2. Certification Credits
3. Limitations and Workarounds
4. Migration Information
5. Supported External Tool Versions

1. New Features and Enhancements

This section describes the new features and enhancements introduced in SCADE LifeCycle 2024 R2 release.

SCADE LifeCycle ALM Gateway

- Support for PTC Codebeamer™ 2.1.0.0
- Capability to remove pending traceability links in SCADE projects

2. Certification Credits

The SCADE LifeCycle 2024 R2 release includes the following tool with certification credits:

- **SCADE LifeCycle Reporter 2021 R2 for SCADE Suite Certification Data Rev B**
 - For qualification under DO-178C (DO-330 TQL-5)
 - For certification under ISO 26262, IEC 61508, and EN 50128
- **SCADE LifeCycle Reporter 2022 R2 for SCADE Display Certification Data**
 - For qualification under DO-178C (DO-330 TQL-5)
 - For certification under ISO 26262, IEC 61508, and EN 50128
- **SCADE LifeCycle Reporter 2020 R2c for SCADE Test Certification Data**
 - For qualification under DO-178C (DO-330 TQL-5)
 - For certification under ISO 26262, IEC 61508, and EN 50128
- **SCADE LifeCycle Reporter 2024 R1 for SCADE ARINC 661 UA DF Generator Certification Data**
 - For qualification under DO-178C (DO-330 TQL-5)

- **SCADE LifeCycle Model Change 2021 R2 for SCADE Suite Certification Data Rev B**

- For qualification under DO-178C (DO-330 TQL-5)
- For certification under ISO 26262, IEC 61508, and EN 50128

3. Limitations and Workarounds

This section describes the limitations and workaround solutions applying to the SCADE LifeCycle 2024 R2 release.

Limitations in Requirements Management

- Exporting models to DOORS NG, Jama, or Polarion is slow (API-related issue from each ALM tool).

Workaround: use custom model export from UI to reduce elements to export and launch export in batch.

- **Codebeamer Connections:**

- If API throttling is active in Codebeamer ALM tool, import/export is slower.

- **Polarion Connections:**

- Filtered documents cannot be imported if work items are referenced from another document (Polarion API issue)
- Impossible to delete exported elements: Polarion API adds a "TO_DELETE" tag on such elements.

Workaround: filter elements tagged by "TO_DELETE".

- **DOORS NG Connections:**

- Selecting views is not available in DOORS Next Generation 6.0.5 & 6.0.6 connection, if Jazz Team Server has no Global Configuration Manager installed.
- When working with DOORS NG 6.0.4, accessing the local configuration of DOORS NG is limited to the content of the initial stream.

Workaround: fixed by using 6.0.5 or higher.

- **DOORS 9.x (native) Connections:**

- When using DOORS baselines for traceability, requirement location by navigation from SCADE projects to ALM tools is not possible.

- **SCADE Suite:**

- It is not possible to trace model objects defined textually in .scade files. [CR110-44]
- Exporting model objects defined textually in .scade files causes object duplication or errors in DOORS, DOORS NG, Polarion, or Jama tools.

- **SCADE Architect:**

- When using default full export, it is not possible to export back to ALM tool databases an AUTOSAR, Avionics, FACE, or any SCADE Architect configured model. Instead use custom model export.

Limitation in Reporter

- **SCADE Architect:** configured models are reported as standard SCADE Architect models and configuration aspects are ignored.
- **SCADE Suite, SCADE Architect, and SCADE Test:** Use of Unicode UTF-8 option in Windows Regional settings is not supported for report generation.

4. Migration Information

ALM Gateway 2024 R1 to 2024 R2

- When exporting models with images, all images (not exported previously due to a bug) are exported at once for the first export using the new release.

ALM Gateway 2023 R2 to 2024 R1

- No format change: no project migration issue

ALM Gateway 2023 R1 to 2023 R2

- ALMGP and ALMGR files format changed to include baselines. On ALMGP, views are automatically migrated to the new format.

ALM Gateway 2022 R2 to 2023 R1

- No format change: no project migration issue

ALM Gateway 2022 R1 to 2022 R2

- ALM Gateway project (.almgp) format change: when opening any SCADE project, users are warned that the ALM Gateway project version is an old version and can be saved into the new format by saving the project. After saving, SCADE projects have no backward compatibility to former ALM Gateway project.
- In SCADE Display/ARINC projects, ALM Gateway projects are now using a single pair of almgmt/almgr files for the entire project instead of one pair of almgmt/almgr files for each specification, symbol, or reference object. In addition, when using Reqtify connection, a single trace file is used instead of a trace file for each specification, symbol, or reference object.

The traceability info contained in SCADE Display/ARINC projects need to be migrated to its new format as follows from the Requirements window after loading your project:

1. Press **Import** to save the ALM Gateway project in the new format and merge all traceability info contained in the project into a single pair of almgmt/almgr files (and also into the single trace file for Reqtify usage).
2. Press **Export** to propagate the new format of traceability info to the ALM tool.

After migration, traceability information in SCADE Display/ARINC projects have no backward compatibility.

For migration information related to previous versions, contact <scade-support@ansys.com>.

5. Supported External Tool Versions

This section describes the external tools supported by SCADE LifeCycle 2024 R2 release.

External Tools	Version
IBM® Rational® DOORS®	9.6 (with DOORS client 9.6.1.7 to 9.6.1.12) or 9.7.2
IBM® Rational® DOORS® Next (formerly DOORS Next Generation)	6.0.4/6.0.5/6.0.6 and 7.0.2
Dassault Systèmes® Reqtify™	2017 to 2022x
Jama Software Jama Connect® (formerly Jama Cloud)	8.24 to 9.8.0
Siemens® Polarion®	18.3 to 2304
PTC Codebeamer™	22.10-SP4 to 2.1.0.0

SCADE Solutions for ARINC 661 Compliant Systems

Release Notes

The following sections present the new features and enhancements for SCADE 2024 R2 Solutions for ARINC 661 release.

1. New Features and Enhancements
2. Certification Credits
3. Limitations and Workarounds
4. Migration Information

1. New Features and Enhancements

This section describes the new features and enhancements introduced in SCADE 2024 R2 Solutions for ARINC 661 release.

SCADE ARINC 661 Widget Library

- Widget library upgraded up to Supplement 9 of the standard: 74 widgets and 8 extensions upgraded. Details are available in the SRS document of each widget or extension.

SCADE Test Automation Framework

- New Server Control Proxy component, integrated in ARINC 661 server, handles instrumentation with SCADE Test Automation Framework
- A661ServerTest.exe, the test version of the server, is now replaced by the Server Control Proxy included in the ARINC 661 server
- Test Automation Framework can be used to script interactions in presence of live UAs with complete DF pages

SCADE UA Adaptor

- Graphical Panel Coupling API with full programmatic capabilities:
 - for listing graphics-side entities, logics-side entities, or existing connections by name
 - for creating/deleting connections
- Graphical Panel Coupling API supporting automated connection workflows and easy management of large embedded display projects

SCADE UA Page Creator

- Capability to set finer grid size when assembling HMLs: new minimum dimension of 1.0 pixel

SCADE LifeCycle Integration

- For details, consult [SCADE LifeCycle Release Notes \(p. 257\)](#).

2. Certification Credits

The SCADE 2024 R2 Solutions for ARINC 661 release includes the following tool with certification credits:

- **SCADE UA DF Generator 6.8:** for qualification under DO-178C/DO-330 up to TQL-1
- See also [Certification Credits \(p. 257\)](#) 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 2024 R2 Solutions for ARINC 661 release.

Limitation in A661 Server Compiled with ANGLE

- On a laptop computer with Intel integrated graphics (driver version 27.20.100.8336 or upper), the server may freeze after few minutes if it is run on this graphics chip. Please contact Ansys SCADE Support in case this issue occurs.

Limitation in SCADE UA Page Creator

- The following widgets have no preview for display in Look Definition Preview: ActiveArea, CursorOver, MapHorz_ItemList, MapVert_ItemList, MapHorz_Container, SizeToFitContainer, ShuffleToFitContainer, TabbedPanel

Limitations in SCADE Widget Library

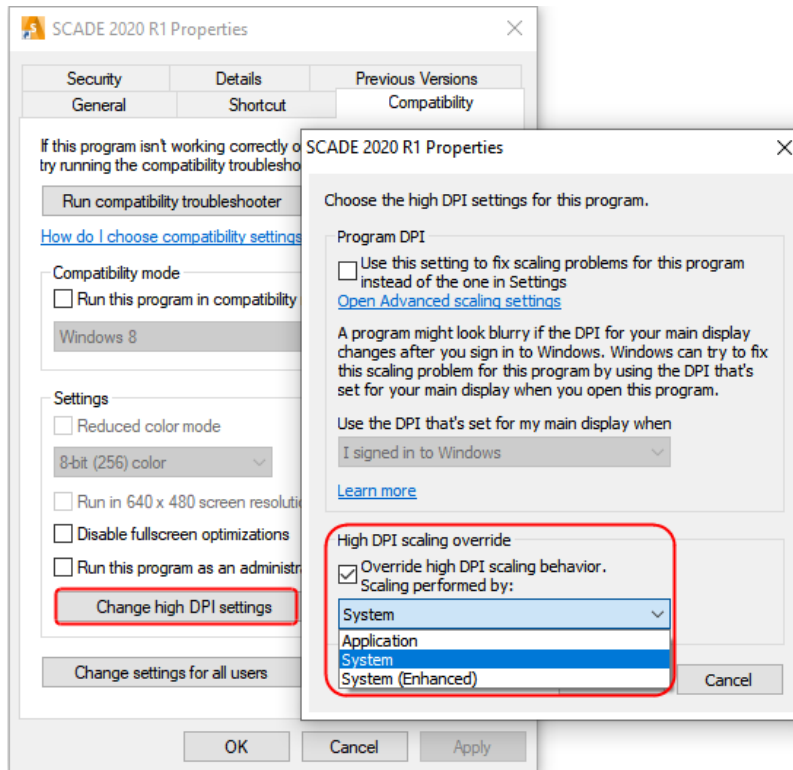
- The A661_REQ_DEBUG_WIDGETS debug widget request and the A661_NOTE_DEBUG_WIDGETS notification are not supported
- Animation escape sequences are not supported in Complex Texts (LabelComplex, ScrollList widgets).
- Animation laws defined in Definition Files are not supported.

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.

4. Migration Information

SCADE 2024 R1 to 2024 R2 Solutions for ARINC 661

- UAPC models using the default A661 Configuration files: automatic migration from A661-6 to A661-9 library format upon loading. For some widgets having modified parameters, their value is lost and the default value is associated with the new parameter:
 - CheckButton: parameter PicturePosition is renamed as CheckboxPosition and is set to A661_LEFT
 - SymbolAnimated: LoopType renamed as LoopFlag and is set to A661_LOOP_FORWARD

For Constants: various categories renamed, constant type is set to uncategorized

SCADE 2023 R2 to 2024 R1 Solutions for ARINC 661

- Format change: at loading, automatic migration of UAPC projects
- Server Creator projects: automatic split of widget capacities/definitions into separate files

SCADE 2023 R1 to 2023 R2 Solutions for ARINC 661

- No format change: no project migration issue.
 - UA Page Creator and Server Creator projects are migrated automatically upon loading

SCADE 2022 R2 to 2023 R1 Solutions for ARINC 661

- UA Page Creator projects are migrated automatically at project loading if they were using the distributed SCADE server
- Server Creator projects need to be redesigned entirely to support new server generation
- New UA Page Creator projects must be created from 2023 R1 Server Creator Project default profile without using profiles created in former versions
- SCADE UA Adaptor: Incompatibility with MapHorz_ItemList and MapVert_ItemList widget connections from previous versions: the common part of the messages is now factorized in arrays. Users must change the SCADE model to match the new structure.
- SCADE UA Adaptor: Incompatibility of A661_REQ_CURSOR_ON_WIDGET and A661_REQ_FOCUS_ON_WIDGET messages connection from previous versions: they are now handled as layer messages as specified in A661 Standard. Users must compute in SCADE model the widget ID for both messages.
- SCADE UA Adaptor: incompatibility of connection between Scade a661_bool enumerate and Emit/Notify message fields from previous version: only Scade bool can now be connected to Emit/Notify fields.

For migration information related to previous versions, contact <scade-support@ansys.com>.

SCADE Avionics Package Release Notes

The following sections present the new features and enhancements for SCADE Avionics Package 2024 R2 release.

- 1. [Fixes and Enhancements](#)
- 2. [Limitations and Workarounds](#)
- 3. [Migration Information](#)

1. Fixes and Enhancements

This release of SCADE Avionics Package is a maintenance release containing fixes and enhancements..

FACE Code Generation

- Capability to generate FACE UoP code from the selection of multiple UoP root operators

2. Limitations and Workarounds

This section describes the limitations and workaround solutions applying to the SCADE Avionics Package 2024 R2 release.

Limitations in SCADE Avionics Package

- Due to CTS known issue, the char "0" should not be used in package naming. CTS checks fail and complain about missing .h/.hpp files if ever using this character in package naming.

Workaround: use CHK_394 to detect and fix invalid naming of concerned elements.

- **IDL Types:** String (and WString in FACE 2.1) are not supported by synchronization with SCADE Suite.

Workaround: use BoundedString instead.

- **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.

3. Migration Information

SCADE Avionics Package 2024 R1 to 2024 R2

- No specific issue: see [SCADE Avionics Package \(each release\) \(p. 266\)](#)

SCADE Avionics Package 2023 R2 to 2024 R1

- No specific issue: see [SCADE Avionics Package \(each release\) \(p. 266\)](#)

SCADE Avionics Package 2023 R1 to 2023 R2

- No specific issue: see [SCADE Avionics Package \(each release\) \(p. 266\)](#)

SCADE Avionics Package 2020 Rx to All Releases

- Existing FACE projects using the SDM delivered as part of SCADE product installation are migrated automatically at project load.
- When using another SDM, the SDM plugin for SCADE must be migrated as follows:
 1. **SDM Migration:** Copy the SDM plugin in a new directory. Then, the SCADE SDM model contained in that directory must be migrated; this is done simply by loading/saving the model with the new SCADE release.
 2. **SDM reference from FACE projects:** using any text editor, in each ETP project file, update the "@SYSTEM:DROPINS" property to the new SDM directory. In the following example, verify the last reference to sdm plugin corresponds to the migrated SDM.

```
<Prop id="55" name="@SYSTEM:DROPINS">  
  <value>$(SCADE)/dropins/face</value>  
  <value>$(USER_PLUGINS_FOLDER)</value>  
  <value>$(SCADE)/dropins/face/sdm/3.1.4</value>  
</Prop>
```

Warning: a project must not reference several SDM plugins. Note: to avoid updating this property, put the SDM directories under the \$(SCADE) variable that expands into the installation path of the current SCADE release as shown in example above. Alternatively, use any user environment variable.

3. **FACE projects can then be migrated in the same way as all SCADE Architect projects:** as explained for each release migration.

SCADE Avionics Package (each release)

- SCADE Architect projects referencing libraries and/or configurations must be properly migrated. The migration of projects with libraries can be automated by a script distributed within SCADE installation and located at \SCADE\scripts\Migration\Architect. For more information, check ReadMe available here and section about "Managing Project Compatibility and Migration" in SCADE Architect User Manual.
- If ever using an in-house configuration extending the FACE or AADL (or Avionics) configuration provided with SCADE Avionics Package, it is necessary to regenerate the in-house configuration and redeploy it within teams using it. If the configuration contains libraries, one has to do the same for libraries, starting from them.
- If using configured projects (based on configurations provided with the product or based on in-house configuration previously updated), it is necessary to load/save all models, starting from libraries if any.

For migration information related to previous versions, contact <scade-support@ansys.com>.