Release Notes

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

Release 2025 R1
January 2025
ANSYS, Inc. and ANSYS Europe, Ltd. are UL registered ISO 9001:2015 companies.

Copyright and Trademark Information

Disclaimer Notice

U.S. Government Rights

Third-Party Software

Published in the U.S.A.

Table of Contents

List of Figures

Global Release Notes

The release notes are specific to Ansys, Inc. Release 2025 R1 and arranged by application or product, with the exception of:

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

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

Archived Release Notes

Archived release notes for releases prior to 2025 R1 can be found in the online version of 2024 R2 release notes . Or via ansys.com> Customer Center> Customer Center> General Support> Download Center> Downloads> Product Documentation> Select Release> PDF packages> release.

1. Advisories

- Ansys Viewer 2025 R1 will be the final version of Ansys Viewer released by Ansys. As of this release, no further updates or bug fixes will be provided.

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

  If you use Ansys Viewer for graphics visualizations, Ansys encourages you to transition to alternative tools or workflows. If you have any questions or need assistance, please contact Ansys support.

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

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

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

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

- The Mechanical application, for Additive Print functionality. Several types of geometry-based AM simulations are available, including laser powder bed fusion (LPBF), directed energy deposition (DED), sintering, and distortion compensation.
- PyAdditive, for Additive Science functionality. PyAdditive allows you to experiment with process parameters for determining the best combination to use in laser powder bed fusion processes of metal parts. See PyAdditive in the Additive Manufacturing Release Notes.

- Ansys Distributed Compute Services (DCS) 2024 R1 was the final version of DCS released by Ansys. Support for DCS 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 Ansys Known Issues and Limitations. 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 2025 R1, refer to the *Resolved Issues and Limitations* document on the Ansys Help site.

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

2. Compatibility with Previous Releases

Backwards Compatibility: Ansys 2025 R1 was tested to read and resume databases from the following previous versions: 2023 R1, 2023 R2, 2024 R1, and 2024 R2. 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 2025 R1, first resume it in a supported version and then resume that database in 2025 R1.

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

3. Installation

The following features are new or changed at Release 2025 R1. Review these items carefully.

- The Ansys installation program has been updated to support RedHat 9, Ubuntu 22, OpenSSL, GCC and ABI=1.
- The Ansys installation program has been updated to support Ansys Speos for NX 2412.
- The Ansys installation program has been updated to support Ansys Material Designer.
- The Ansys installation program has been updated to remove Ansys SpaceClaim.

4. Licensing

The following enhancements were made to Ansys, Inc. Licensing for Release 2025 R1:

- The Ansys License Manager has been updated to FlexNet Publisher client version 11.19.5.0 and server version: 11.19.5.1.
- The Ansys License Manager has been updated to OpenSSL version: 3.0.14.
- The Ansys License Manager has been updated to Java version: 17.0.13.11.
- The Ansys License Manager has been updated to Tomcat version: 10.1.28.
- The Ansys License Manager supports user specific license settings that enable users to change settings without administrator privileges.
- License server logs have been improved to include helpful diagnostic information.
- Shared Web has added access to several Ansys Autonomy, Electronics, Fluids, Optical, Structures and Semiconductor products.
- The Ansys Licensing Portal has added new filtering functions to the reports functionality.
- The Ansys Licensing Portal Web Transactions Table now shows denials in the Status column and additional information about the denial in the Detail column.
- There have been extensive upgrades to the appearance and capabilities of the Ansys Licensing Portal tables.
- Customized table views in the Ansys Licensing Portal are now automatically saved and restored.

5. Documentation

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

Ansys Help

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

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

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

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

6. Verification Manuals

The Verification Manuals for the following products were updated for 2025 R1:

6.1. Mechanical APDL Verification Manual

6.1.1. New Test Cases

- VM322: **J-Integral Calculation for Crack in Finite Deformation-Finite Strain Hyperelastic Solid**: An infinitely long strip of a thin rubber sheet is clamped at the top and bottom edges with an applied displacement. An incompressible Neo-Hookean material model in plane stress is used. J-integral values are determined. The Mechanical APDL results are then compared the reference.

- VM323: **Total Strains of a Bar with Combined Plasticity and Creep**: A bar is modeled using BISO material and a time-hardening creep model. Uniaxial pressure loading is applied in the vertical direction. The loading history is composed of three steps. The total strain history is obtained and compared against the reference solution.

6.1.2. Modified Test Cases

- VM97 was modified to include four-node thermal shell element SHELL294 (MEMBRANE).

6.1.3. Removed Test Cases

No test cases were removed for 2025 R1.

6.2. Workbench Verification Manual

6.2.1. New Verification Test Cases

No test cases were added for 2025 R1.

6.2.2. Modified Test Cases

No test cases were modified for 2025 R1.

6.2.3. Removed Verification Test Cases

No test cases were removed for 2025 R1.

6.3. Ansys LS-DYNA Verification Manual

Test Case Sets:

6.3.1. LS-DYNA SOLVE Test Cases

The following test cases were added for 2025 R1:

- VM-LSDYNA-SOLVE-034: **Evolution of the Solder Ball Shape Using 1521 Particles**: This test case validates the solder standoff height of a solder ball using 1521 particles by reproducing the evolution and final shape of the solder.
- VM-LSDYNA-SOLVE-035: **Evolution of the Solder Ball Shape Using 2197 Particles**: This test case validates the solder standoff height of a solder ball using 2197 particles by reproducing the evolution and final shape of the solder.
- VM-LSDYNA-SOLVE-036: **Adiabatic Expansion of High Explosive Detonation Products Using an Eulerian Approach**: A copper cylinder containing TNT explosive is detonated at the base using a planar detonation wave. The radial expansion of the copper cylinder is measured.
- VM-LSDYNA-SOLVE-037: **Adiabatic Expansion of High Explosive Detonation Products Using a Lagrangian Approach**: A copper cylinder containing TNT explosive is detonated at the base using a planar detonation wave. The radial expansion of the copper cylinder is measured.

6.3.2. LS-DYNA EMAG Test Cases

No test cases were added for 2025 R1.

6.3.3. Modified LS-DYNA Test Cases

No test cases have been modified for 2025 R1.

6.3.4. Removed LS-DYNA Test Cases

No test cases have been removed for 2025 R1.

7. Online Video Access

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

8. Ansys Customer Site

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

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

Ansys customer site access points:

- **Tutorials and input files**

  To access tutorials and their input files, go to the tutorials area of the customer site.

- **Documentation**

  To access documentation files, go to the documentation area of the customer site.

- **General information**

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

Ansys 3D Design Products

Release Notes

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

Discovery Release Notes

Release 2025 R1 of Discovery contains the following.

1. Discovery Product Advisories

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

Installation and Licensing Advisories

- Support for Microsoft Windows 11 version 21H2 has been dropped.
- 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, see Installation Prerequisites.
- Discovery has moved to a Pro, Premium, Enterprise (PPE) licensing scheme. The former Discovery Modeling is now Discovery Pro, the former Discovery Simulation is now Discovery Enterprise, and a new mid-tier offering, Discovery Premium, is now available. Also see Installation and Licensing (p. 27).
- If you are installing Discovery as a component of another Ansys product package, such as an Ansys Fluids or Ansys Structures package, you may now choose between a full or reduced Discovery installation. The reduced option is suitable for preprocessing only. It does not include the Fluent or MAPDL solvers, thereby limiting the use of the Refine stage as well as impacting physics transfer downstream. Also see Installation and Licensing (p. 27).
- The **Discovery Licensing Mode Manager** has been retired.
- The **Legacy Named User Subscription** option is no longer supported. It has been replaced with web-based Named User licensing.

Geometry Kernel Advisories

The underlying geometry kernel for SpaceClaim has changed from ACIS to Parasolid. Behaviors you can expect due to this change include:

- Large unit support for *.scdoc files that are outside of bounding box > 1000 m has been added to Discovery.

Feature Removal

- The `--use-legacy-translator` command line option for Discovery has been removed.

SpaceClaim Advisories

- The underlying geometry kernel for SpaceClaim has changed from ACIS to Parsolid.
- SpaceClaim 2024 R2 was the last version based on ACIS.
- API version V22 has been deprecated and will be removed at Release 2025 R2.

  If you have customized add-ins or scripts, make sure they are upgraded to the most current version.
- SpaceClaim files are imported using a translator. Opening a SpaceClaim file may take longer than opening a Discovery file due to the geometry conversion.
- SpaceClaim has been removed from the Ansys unified installer and the standalone Discovery product installation, but you can still install SpaceClaim using any of these methods:
  - Download the SpaceClaim standalone installation package from the Customer Portal and install it
  - Use the command line option **-install_spaceclaim**
  - Automatically as a component of a Material Designer or Speos installation

  See SpaceClaim Installation and Licensing for details.

Discovery Beta Features

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

Also see Discovery Enhancements Overview (p. 4) for features that have been added or enhanced.

2. Discovery Enhancements Overview

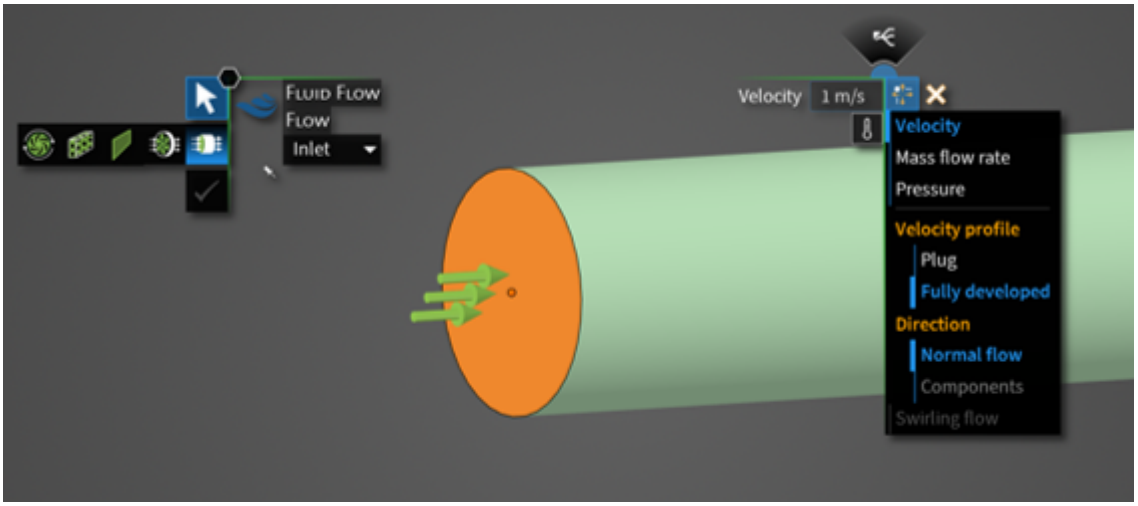
For product advisories and changes to Discovery features that existed in earlier releases, see Discovery Product Advisories (p. 3).

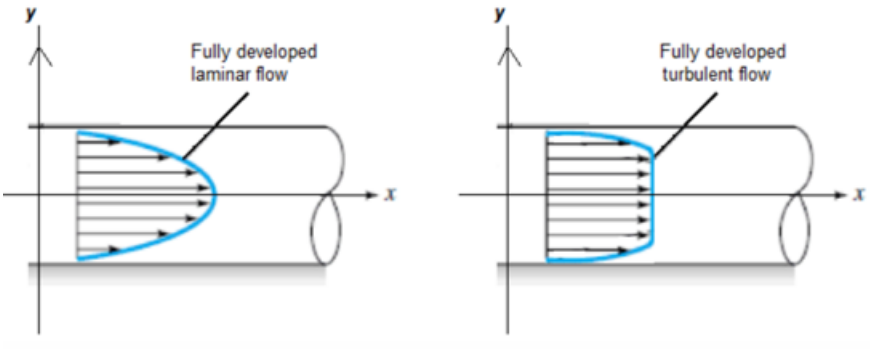
Areas where you will find new capabilities and enhancements in Discovery include:

2.1. Fluid Flow Simulation

2.1.1. Velocity Inlet Profiles

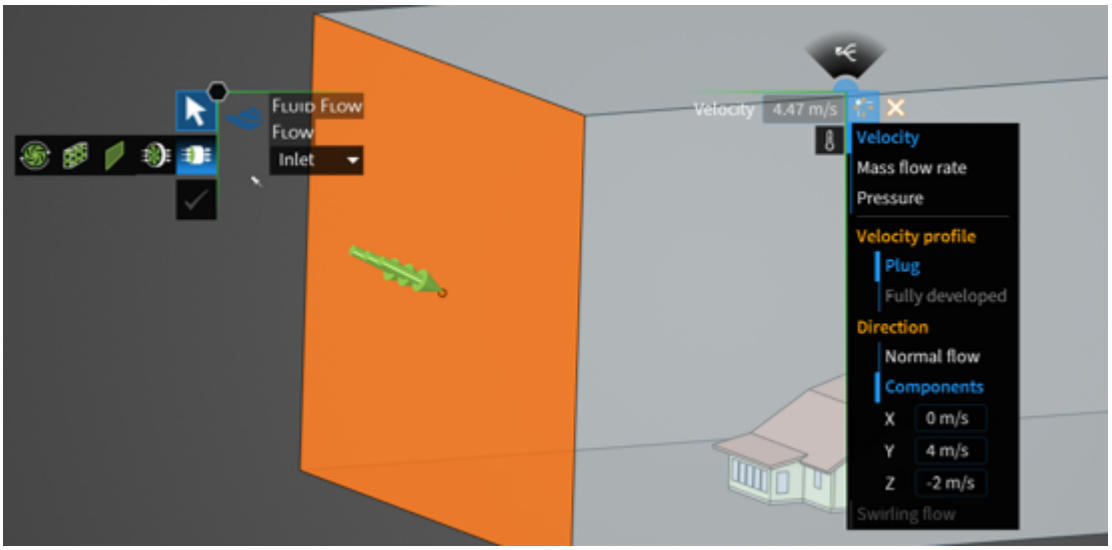
A new option is available to specify a fully developed profile (in **Refine**) for circular velocity inlets. The laminar or turbulent inlet profile is determined from the turbulence model.





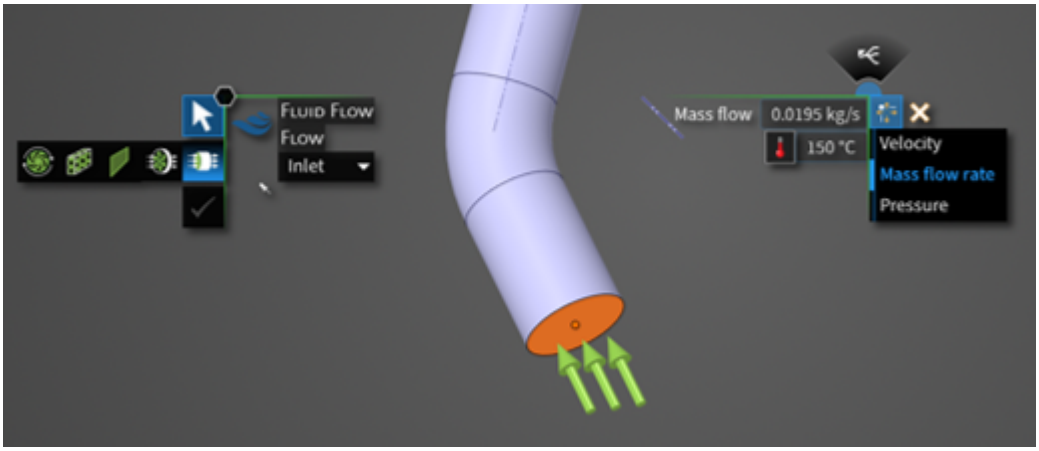
2.1.2. Velocity Inlet Flow Components

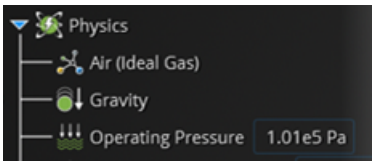
A new option to specify the X, Y, and Z components for velocity inlet is available. It provides additional control for defining the inlet flow direction.



2.1.3. Mass Flow Rate

Mass flow rate specification at a flow inlet is now supported in compressible flow simulations in **Explore**. The new capability to combine mass flow inlet with the ideal gas law provides additional flexibility for defining flow conditions for subsonic gas flows or flows with large temperature variations.



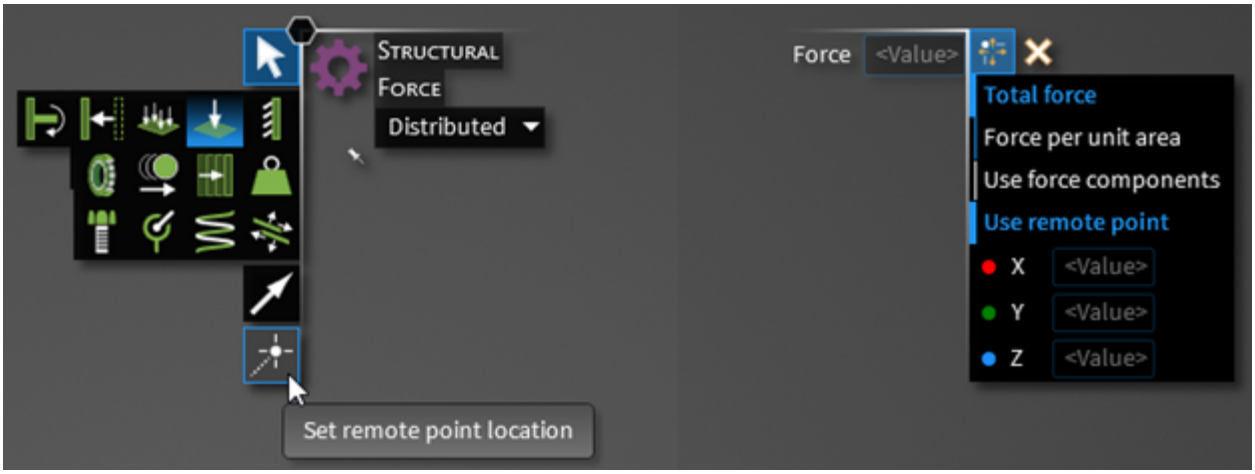


2.2. Structural Simulation

2.2.1. Define Remote Points by Geometry Selection

You can now select a point, edge, face or body to define remote points for structural conditions, including force, mass, and displacement. The centroid of the geometry selection defines that remote

point location. This option improves the workflow for the remote force, remote mass and remote displacement conditions.



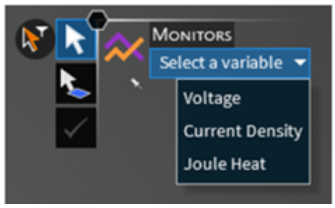
2.3. Electromagnetic Simulation

2.3.1. New Voltage and Current Conditions

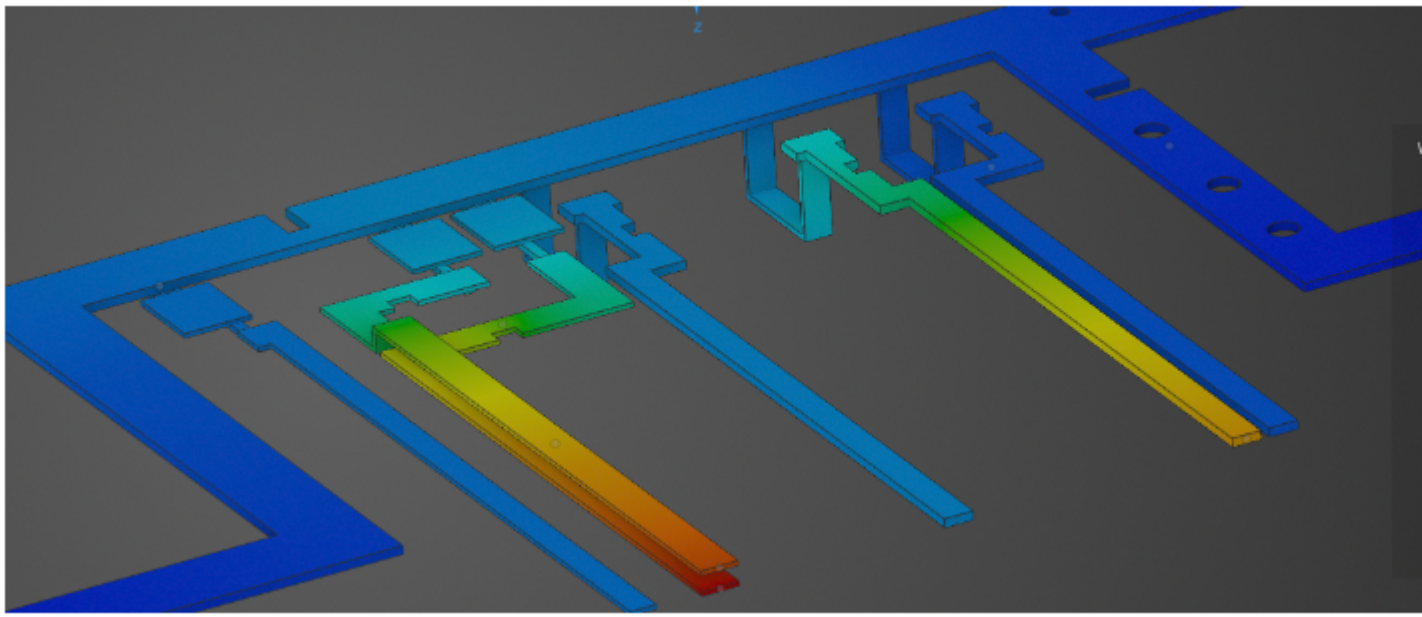
New conditions for voltage and current were added. Voltage and current can only be applied to faces. Circuit and mode port are disabled when voltage/current is selected.

To add voltage or current: select one or more faces. Press Enter.

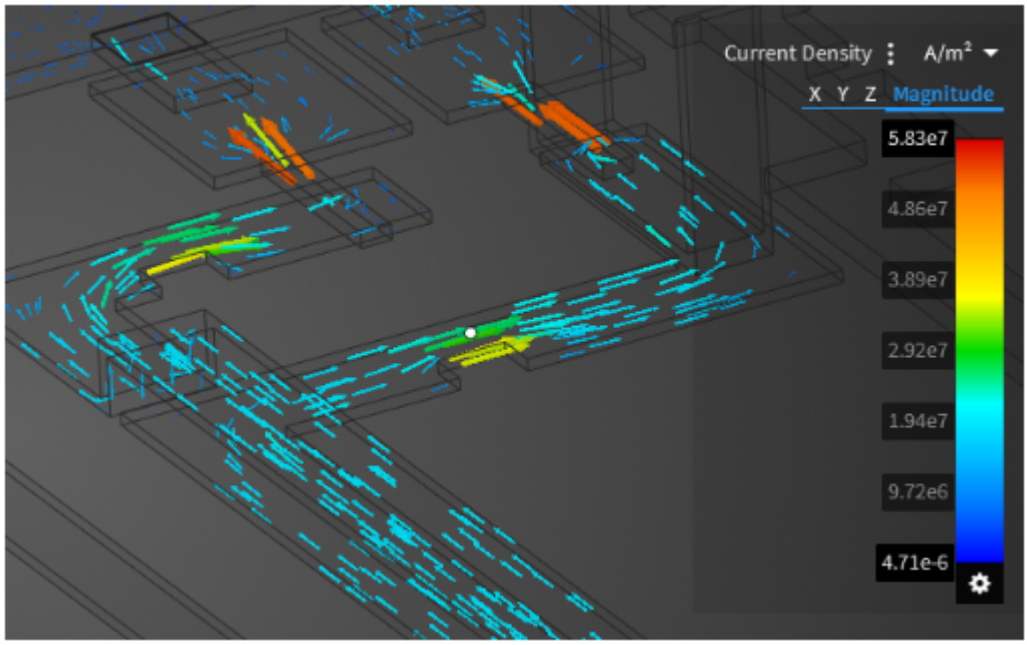
New monitors for electric simulations were added for **Voltage**, **Current Density** and **Joule Heat**.



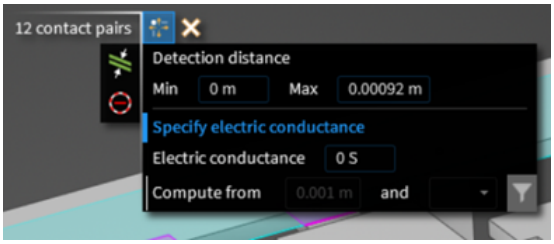
New contours were added for electrical simulations to display **Voltage**, **Current Density**, or **Joule Heat**.



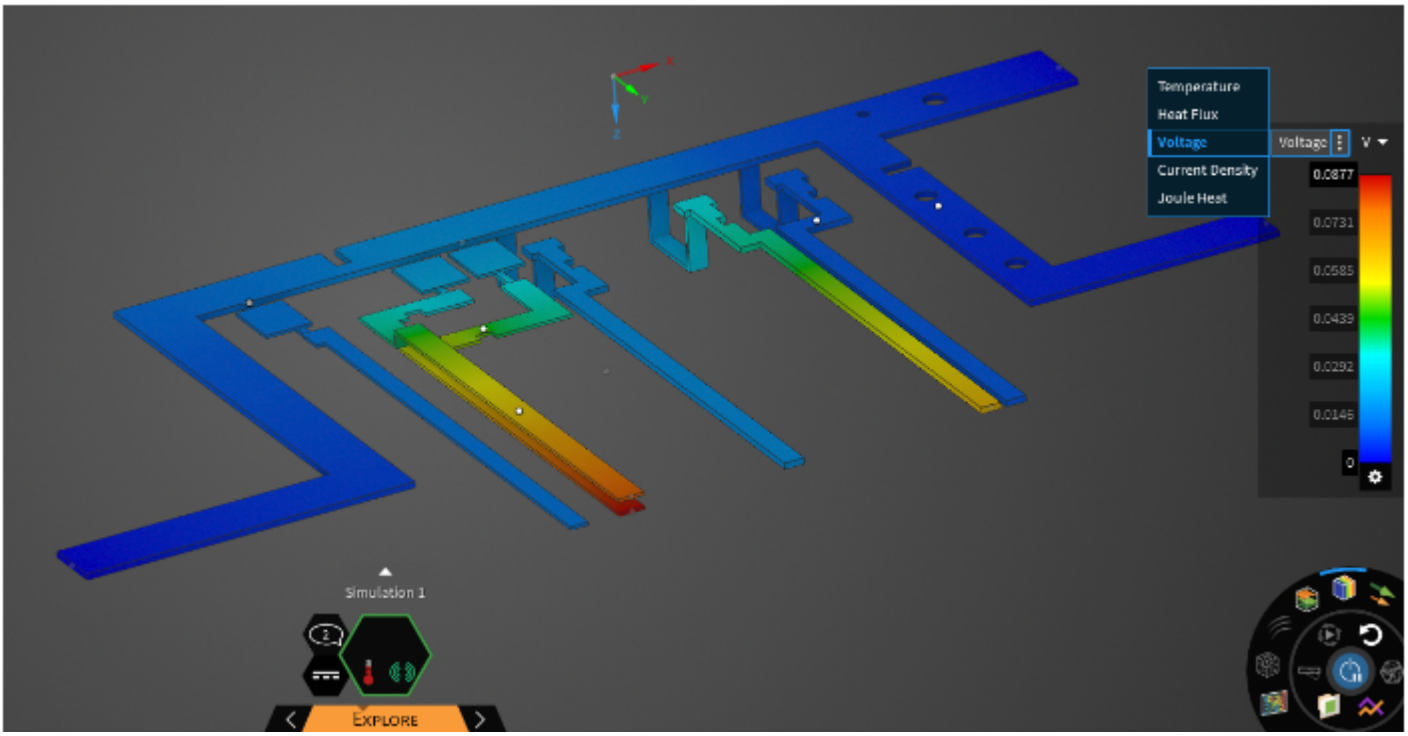
New vectors were added for electrical simulations to display **Current Density**.

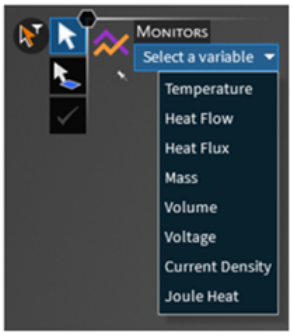


Electrical Conductivity were added as a new material property for Electric Simulation. Additionally, Electrical conductivity is available to be user-defined on contacts using the Contact Review Tool.

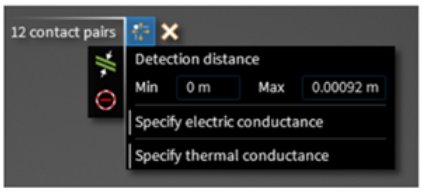


The recently introduced Electric conditions can be used along with the existing Thermal conditions. The integrated Thermal-Electric Simulation provides settings, monitors, contours, and various results visualization options from both the existing Thermal Simulation features and the new Electric Simulation features





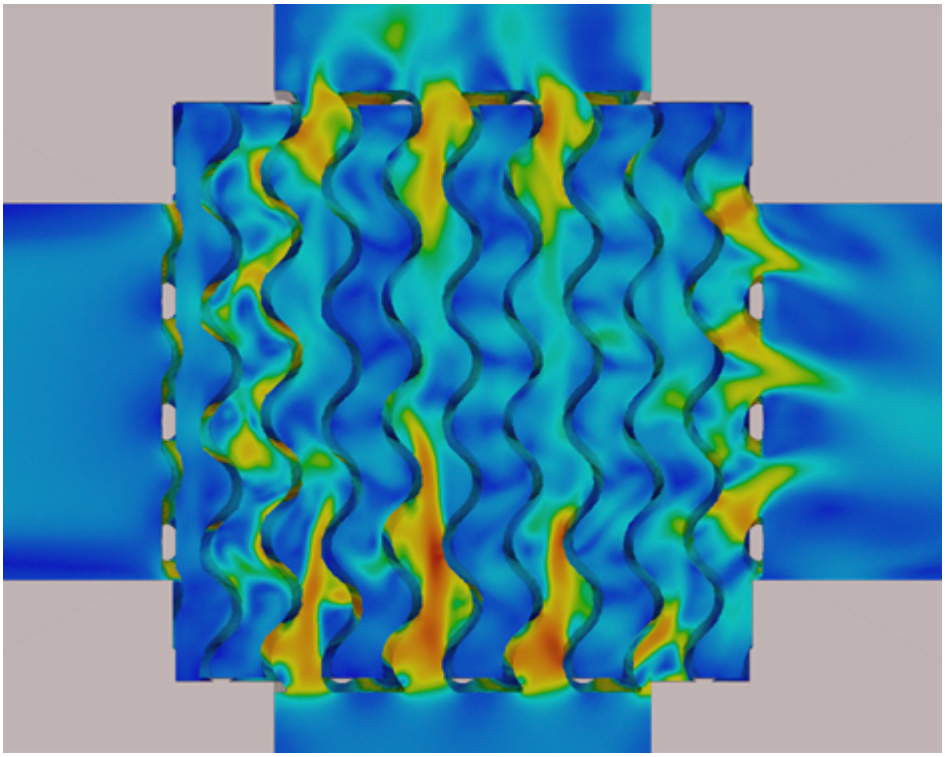
For Thermal-Electric Simulations, both Electrical conductivity and Thermal conductivity are available to be user-defined on contacts using the Contact Review Tool.



2.4. Contacts and Interfaces

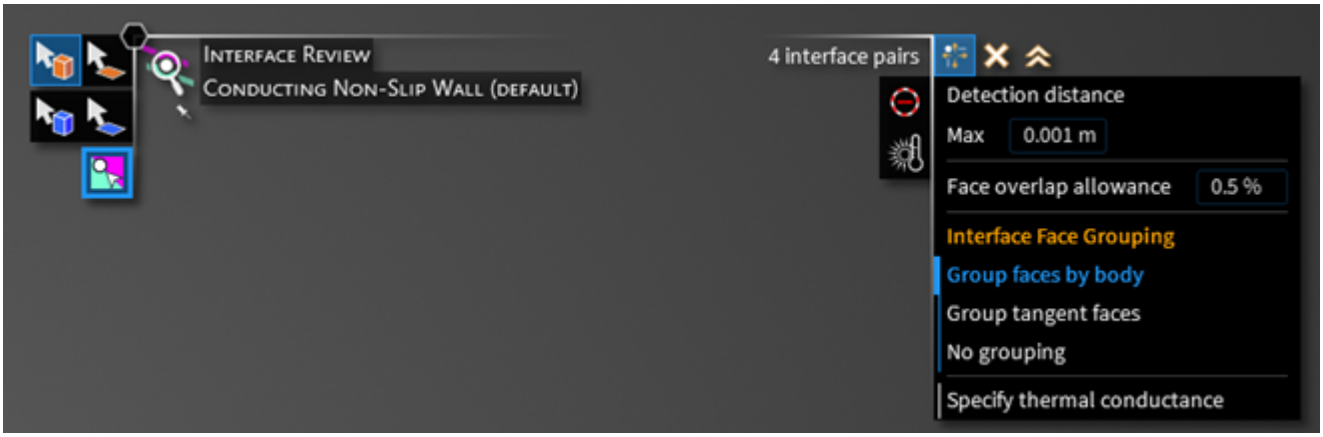
2.4.1. Fluid-Solid Heat Transfer with Faceted Geometry

In **Explore** you can now simulate fluid-solid (conjugate) heat transfer using faceted geometry with automatic interface detection. This new capability enables the evaluation of fluid and thermal performance of high efficiency gyroid heat exchangers.



2.4.2. Face Overlap Allowance

You now have the option to specify a Face overlap allowance for fluid-solid interfaces, useful when you have very small contact points, especially near curved edges. This option defines the percentage that faces are allowed to overlap during interface detection.

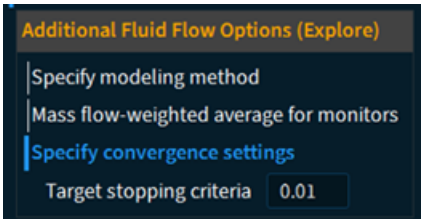


The default for this field is set in the Physics Settings.

2.5. Solution and Optimization

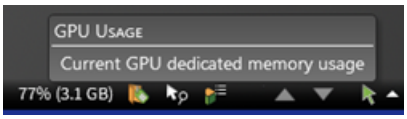
2.5.1. Fluid Flow Convergence Settings in Explore

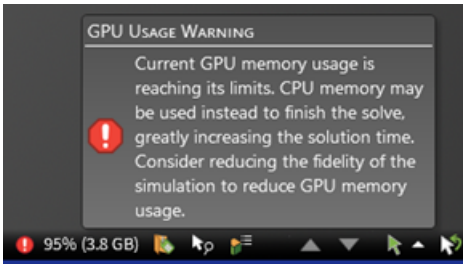
In **Explore**, you can specify the convergence settings by entering the **Target stopping criteria**. Solution convergence is achieved when the percentage change in the solution variables is less than the specified tolerance. A default value of 0.01 (or 1%) is used in the application.



2.5.2. GPU Memory Usage

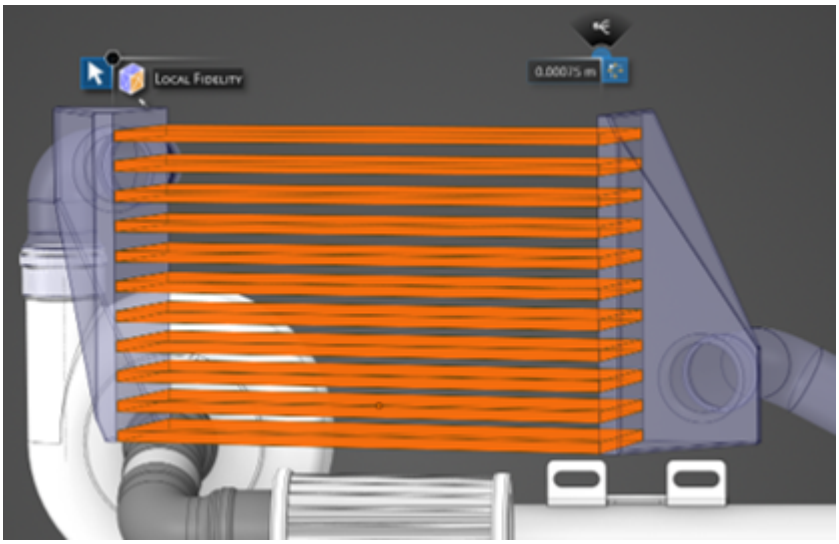
The ability to monitor your GPU memory usage while solving is now available in **Explore**. Additionally, for fluid flow simulations, you can monitor GPU memory usage in **Refine** when using the LiveGX GPU solver. Discovery displays the current GPU memory usage in the status bar. It will also warn you if your current simulation exceeds your GPU's available memory. Excessive GPU memory usage can lead to instability or dramatically slower solve times due to paging to system memory.



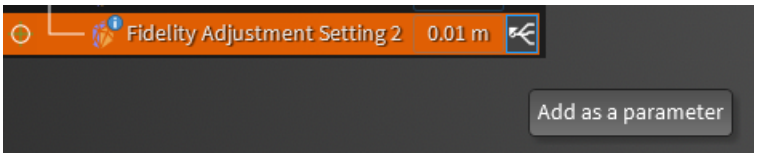


2.5.3. Local Fidelity

Local fidelity for fluid flow simulations in **Explore** can now be applied to bodies, including cutter bodies, as well faces. As a result, thin fluid channels and small components are easier to capture.

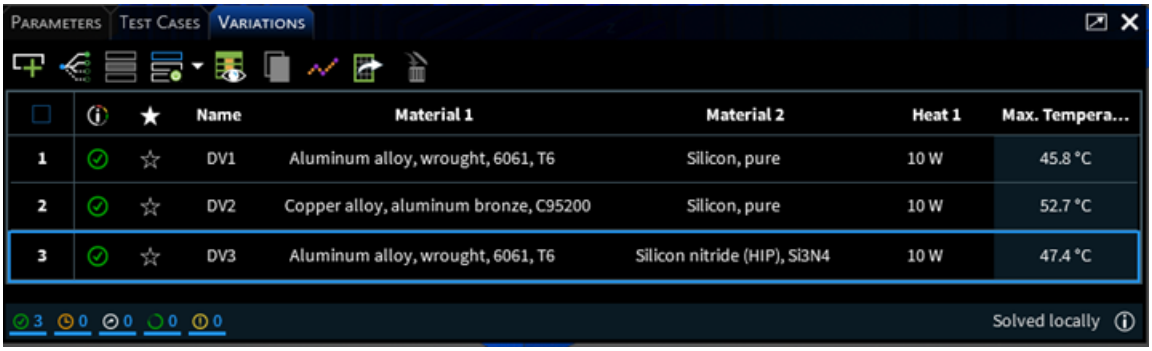


Defined variations and parameter studies can now be applied to local fidelity inputs.



2.5.4. New Materials Input Parameters

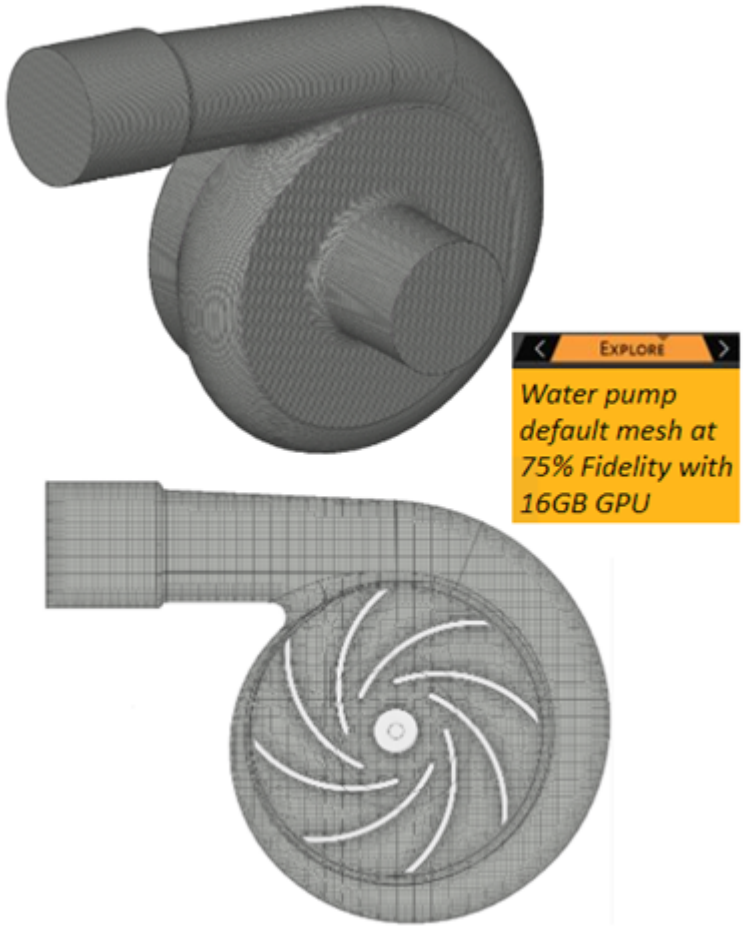
Defined variations can now include materials as input parameters.



2.6. Meshing

2.6.1. GPU Meshing

In **Explore**, GPU meshing is now the default meshing algorithm for all fluids and fluid-solid heat transfer simulation with or without local fidelity. Additionally, fluid simulation in **Explore** now includes the ability to apply a local fidelity on bodies, which can be combined with local fidelity on faces. The new GPU meshing algorithm combined with local size controls makes it easier to discretize thin fluid channels and small components, while simultaneously improving accuracy with a more efficient use of GPU memory.



2.7. Modeling Tools

Note:

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

2.7.1. 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.

2.7.2. Detect Tool

Clearance Detect. The upgraded Clearance Detect feature now operates faster to more effectively pinpoint gaps within the model.

Group Named Selection. A new tool option to **Group named selection** was added to the **Detect** tool. This option is only used if you choose to create a named selection from the HUD. Select it to create a single named selection for the detected areas. Deselect this option to create a named selection per detected area.

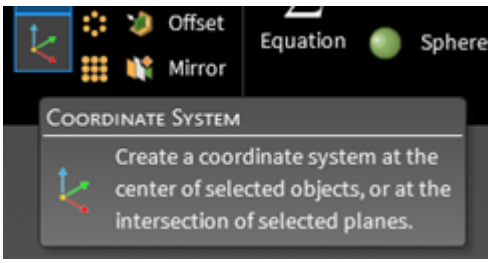
Detect Sweepable Bodies. The Detect HUD now includes a tool for detecting bodies that can be swept. Sweepable bodies have free (unstructured) source and target faces opposite each other and mapped faces on the perpendicular sides between. They can be filled with an all hex and/or prism mesh that is interpolated between the source and target faces. This feature helps you prepare your model for hex meshing and reduces the need to return to Discovery from a downstream tool.

2.7.3. Modeling Tools Additions and Changes

Additional options are now available for the following modeling tools:

- The **Extend** tool in the **Prepare** tab has a new **Create body** option. It will create a weld body between two surfaces.
- The **Enclosure** tool in the **Prepare** tab has a new **Subtract from enclosure** option.
- The **Point** tool in the **Design** tab allows you to Use precise location.

In the **Design** tab, **Origin** was renamed to **Coordinate system**.



2.7.4. Share Topology

The performance of the **Share** tool has been improved at Release 2025 R1.

- Use to preview coincident topology based on the coincidence tolerance and other options selected.
- You can now directly share topology without previewing the coincident topology. This improves performance especially for large models where the preview may consume significant time.

2.7.5. Beams

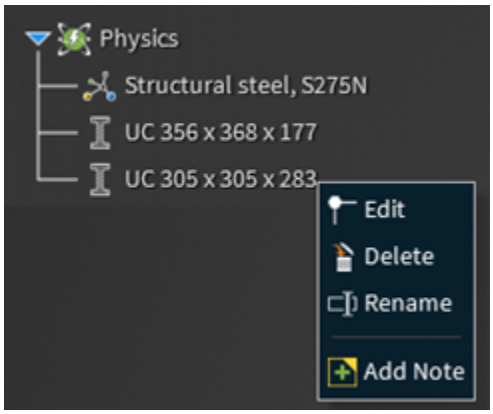
The Beams feature now includes the following new functionality to expand and ease the specification of beams:

2.7.5.1. Create Beams on Body Edges

You can now select bodies, faces or edges to assign beams. The Discovery application will create curves on all edges of selected geometry and assign beams.

2.7.5.2. Remove a Beam Profile

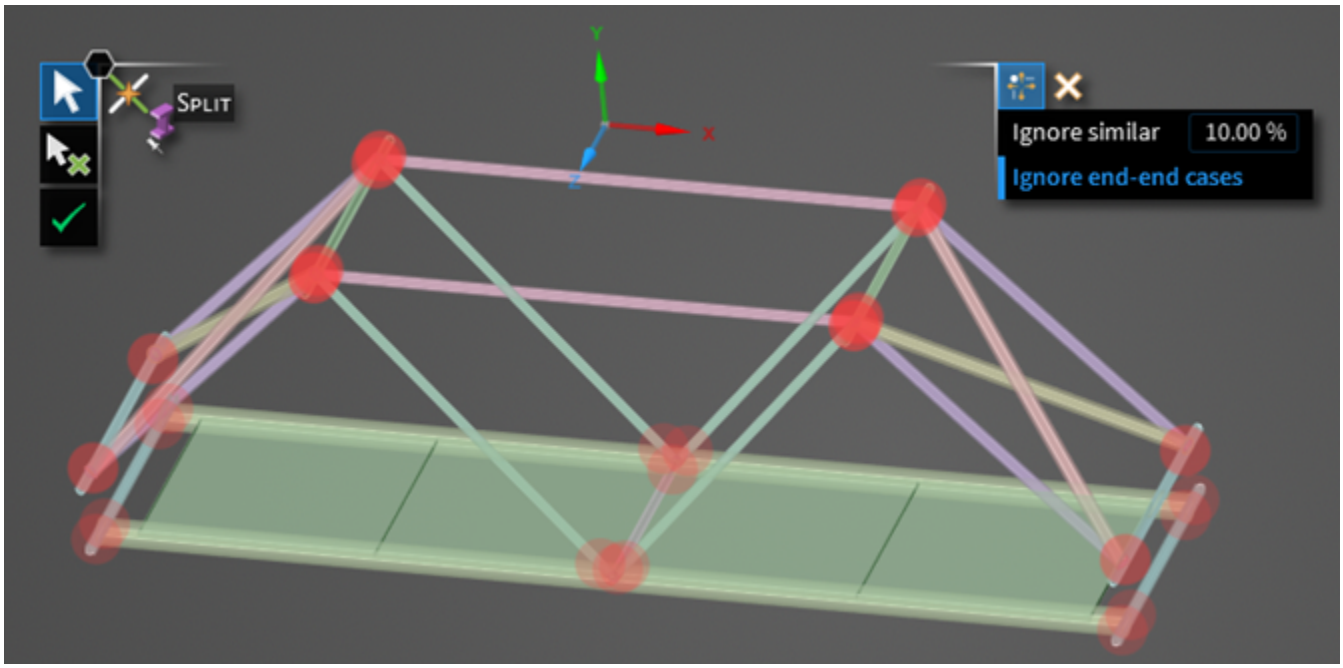
You can now remove a beam profile by right-clicking it in the physics tree.



2.7.5.3. Split Beams

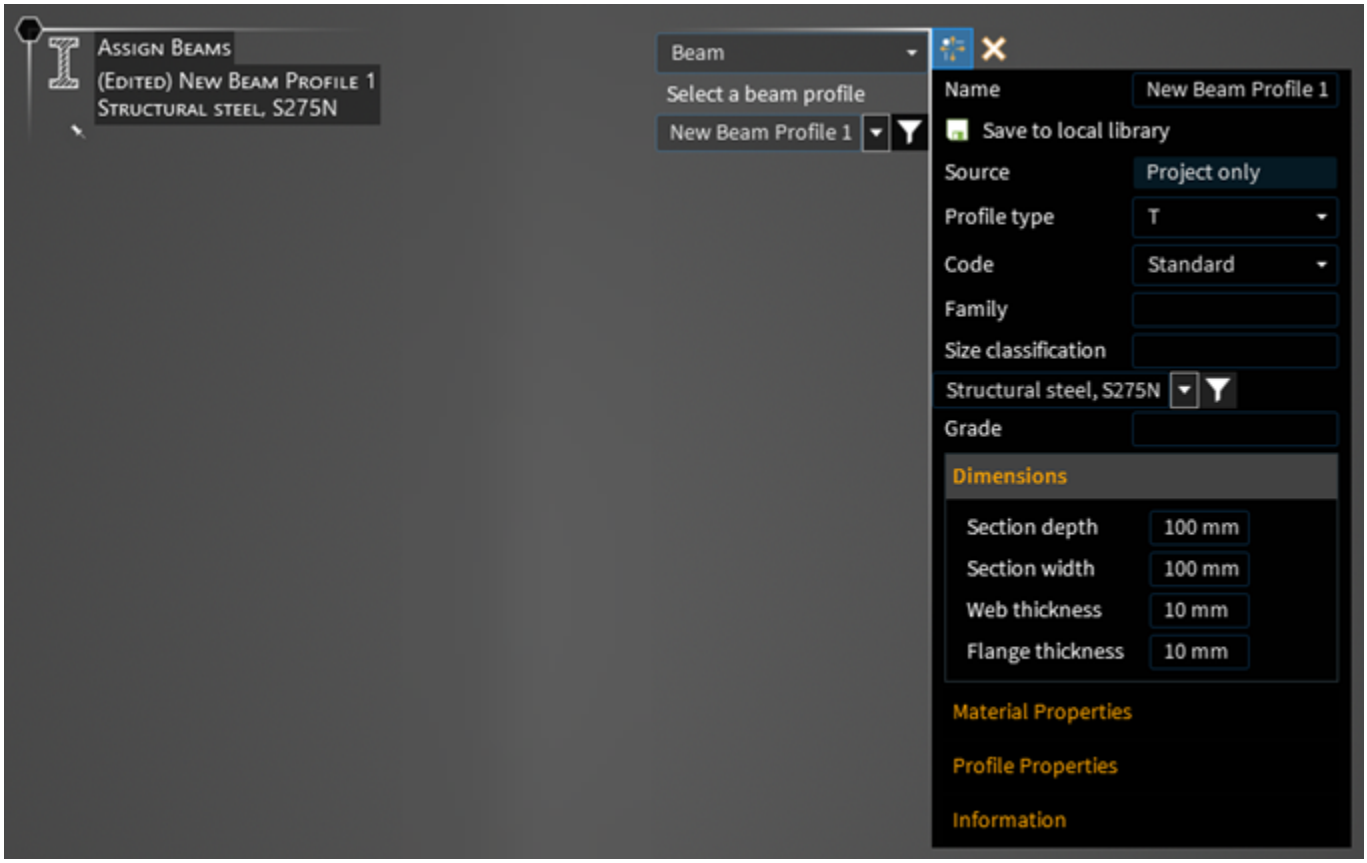
You can now use the **Split** tool to find connections between solid (extracted) beams, where the end of one beam lies somewhere in the middle of another beam.

For any connections found, the beam with the smaller cross section is split by the beam with the larger cross section. Beams with the same cross section are not highlighted. Cross sections that are not the same but are of similar size are also disregarded. You can specify the percentage size difference used to consider any two beams to be similar. You can also choose to ignore beams that meet at end points.



2.7.5.4. Custom Beam Definitions

You can now create a new beam definition, rather than relying on standard library definitions. You can either select and then edit a standard library definition, or select a beam type and modify default values populated based on the selected type.

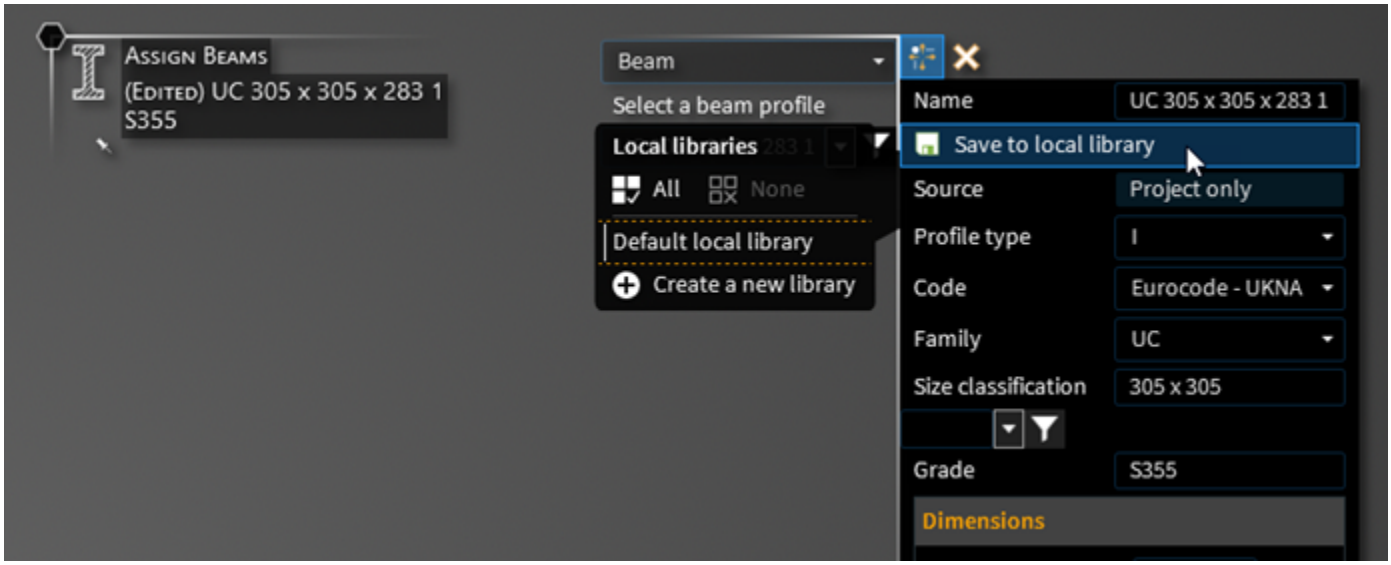


2.7.5.5. Beam Types

You can now select a type for your beam. Types include Beam, Cable, Spring, Link/Truss, Pipe, and Thermal Fluid. The type can be applied to any beam. This attribute is then passed on to downstream applications.

2.7.5.6. Local Library Support for Beams

Once you have created a custom beam definition, you can save it from a local library. Custom definitions are then available for other projects. The local library folder is set in the Physics Settings.



2.8. Sheet Metal

The Sheet Metal tab in Discovery contains tools for working with sheet metal designs.

You can:

- Create or import sheet metal designs.

- Sketch new sheet metal parts and edit or modify existing sheet metal designs.
- Add sheet metal features such as bends, tabs, or hinges to your design.
- Unfold the sheet metal component and view the unfolded component in a new component tab as a top view with its overall dimensions.

Sheet metal settings are available for setting defaults. You can additionally modify the thickness, inner radius, and other sheet metal properties in the **Properties** tab of the Advanced Selection window.

2.9. Materials

The following enhancements related to materials are available in Discovery 2025 R1.

2.9.1. Materials Data for Simulation

Materials Data for Simulation (MDS) is available through an optional license that provides access to simulation-ready materials data for a broad coverage of materials classes, enabling engineers to create more accurate simulations with trusted data. For easy access, MDS is embedded directly into 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 2025 R1 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 2025 R1 release of MDS, but some data values have been updated.

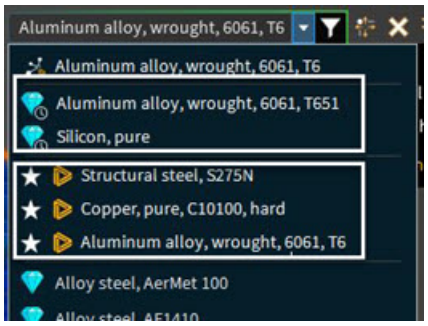
2.9.2. Materials Data for Simulation - Sample Library

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

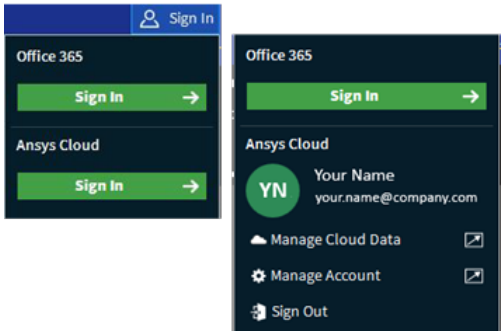
There are no new materials in the 2025 R1 release of MDS, but some data values have been updated.

2.10. User Interface Enhancements

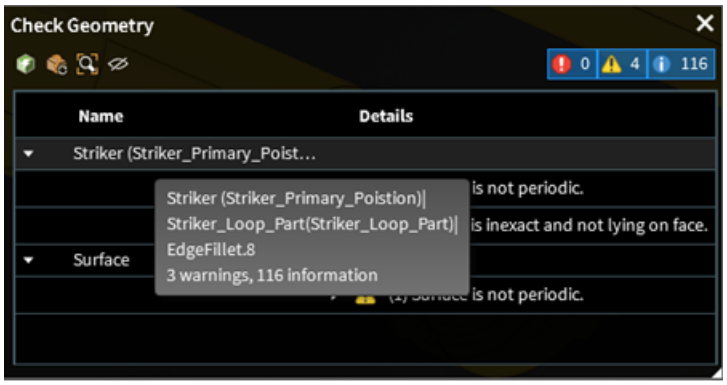
Some dropdown lists that contain many items now display favorites and recently used at the top of the list to improve usability.



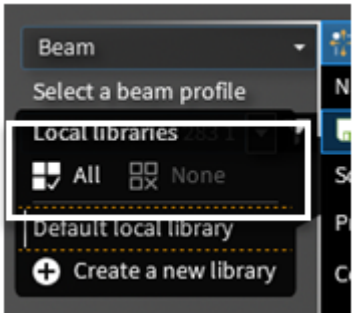
A new sign-in panel includes access to Microsoft 365 as well as Ansys Cloud. If you are logged into Ansys Cloud, you can access the cloud portals to manage your cloud data and cloud account.



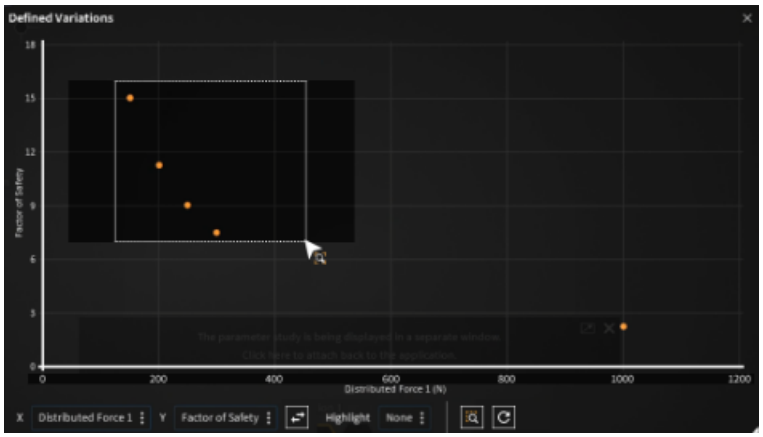
The **Check Geometry** panel truncates long names but the full name is displayed in a dynamic tooltip along with a count of associated errors, warnings or informational messages.



Some assignment tools allow advanced selection with All or None.



With **Zoombox** you can box-select an area on variation charts to zoom in and examine it closely. When you're done, click **Reset Zoom** to return to the original view.



Additionally,

- Changes to visual padding may improve the legibility of visual elements for some users.
- Improvements to messages and notifications continue in this release.
- Improved drag-and-drop file functionality has a larger drop area and greater flexibility when opening or inserting files.

2.10.1. Tree Improvements

Tree improvements include:

- The trees can now be moved out of the main UI.
- Reordering of some nodes within the model tree is now possible.
- Cut plane node now differentiates with naming and different icons between a plane and a cut plane created by Cut plane in the Results arc.
- A lock icon shows on the component parent node when all subcomponents are locked.
- A feather icon displays on the component parent node when all subcomponent bodies are lightweight.

2.11. User Settings Summary

2.11.1. Licensing

Changes and additions have been made to Licensing settings, which were moved from Advanced to their own Licensing section:

- **Discovery license**: You now choose **Discovery Enterprise**, **Discovery Premium**, or **Discovery Pro**.
- **Check for lower license if requested is unavailable**: Select this new option if you want to check for a lower level license if the license requested by the **Discovery license** option is unavailable.

- **Alternate license**: Select an Alternate license for flagship products.
- **Check for Discovery Pro license prior to alternate license**: Select to check whether you have a Discovery Pro license before checking for an alternate license. This option is applicable only when **Discovery license** is set to **Discovery Pro**, and an **Alternate license** preference is specified.

Also see Installation and Licensing (p. 27) for broader changes related to licensing.

2.11.2. Physics

In the Physics Settings, you can now specify a default for the **Face overlap allowance** in fluid-solid interfaces. For more information, see Face Overlap Allowance (p. 10).

2.11.3. Results

Generate default scenes: You can control the generation of default physics saved scenes globally through the Settings panel. There you can also set the default view orientation for scene capture. Options include home, isometric, trimetric, right, left, top, bottom, front and back.

2.11.4. Sheet Metal

A new section was added for sheet metal settings. Set **Basic** and **Relief** defaults, settings for **Export**, and for **Lines** and **Colors** for **Layers** and individual sheet metal elements.

2.11.5. Advanced

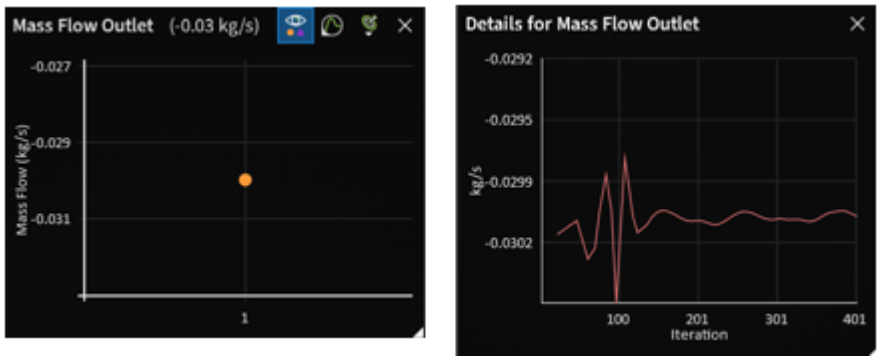
Maximum undo steps: Undo/redo improvements include a scrollbar to navigate the undo/redo list. The number of items in the undo/redo list is customizable but the maximum will depend on local display settings. Set the **Maximum undo steps** in Advanced Options on the Advanced tab of the Settings panel.

2.12. Results

2.12.1. Monitors

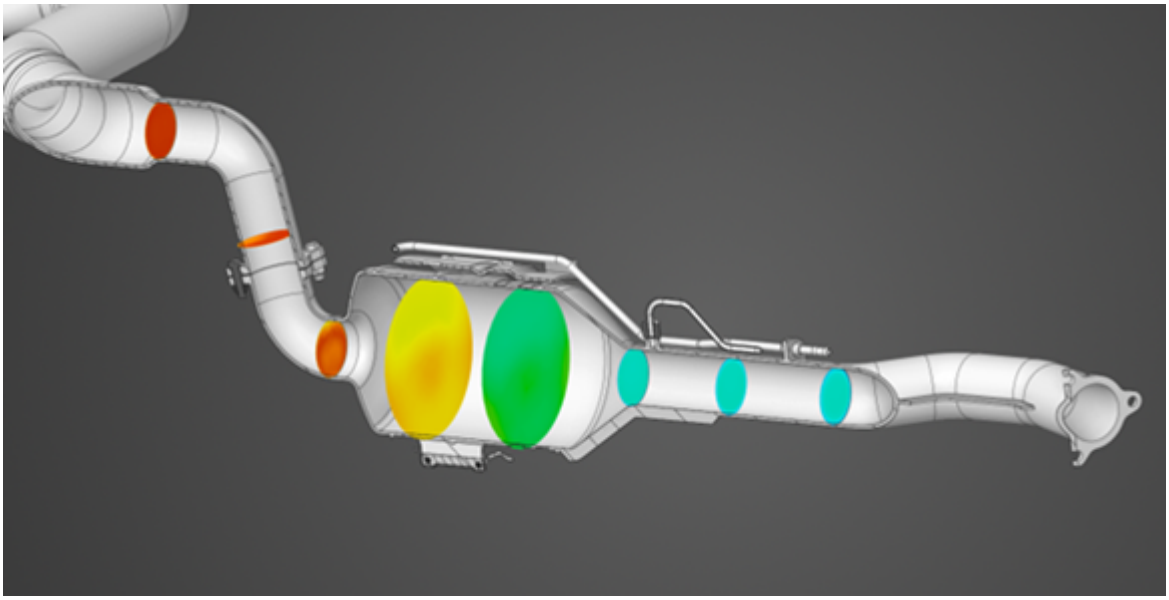
Detail monitors and standard monitors are now based on node values. Prior to this release, the detail monitor value was a face-based result while the standard monitor value was a node-based result.

Detail monitors are now available in **Explore** for reviewing engineering quantities of interest as a function of solution iteration. Use monitor details to judge solution convergence in steady-state fluid flow and fluid-solid heat transfer simulations.



2.12.2. Contours on Planes

Enhancements have been made for displaying contour results on planes, which improve the speed of display and robustness. Contour results on planes are shown in Explore and Refine modes for all physics except electromagnetics. Planes for LiveGX are also supported, with results being updated as the geometry changes.

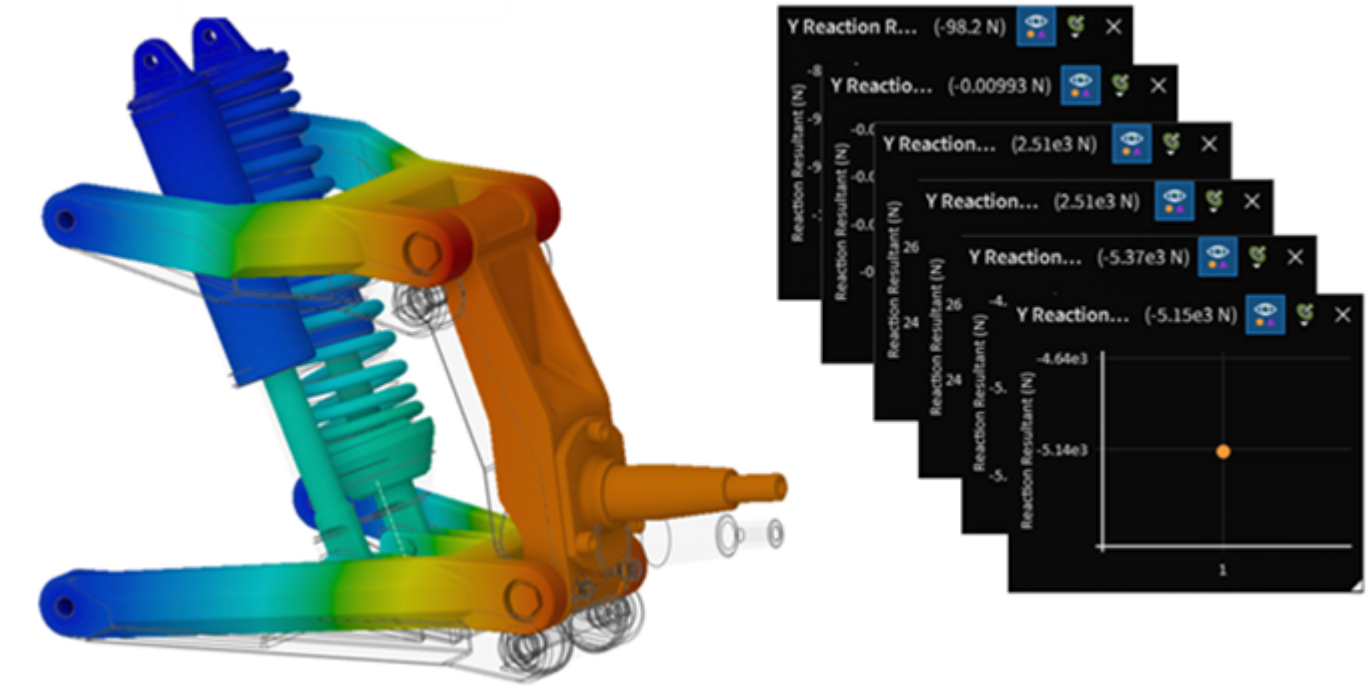


2.12.3. Default Saved Scenes

Saved scenes has expanded functionality to generate scenes automatically for most physics conditions. You can control the generation globally through the **Settings** panel or, for a given document, through the **Scene** dropdown list in the Simulation tab. You can also set the default view orientation for scene capture.

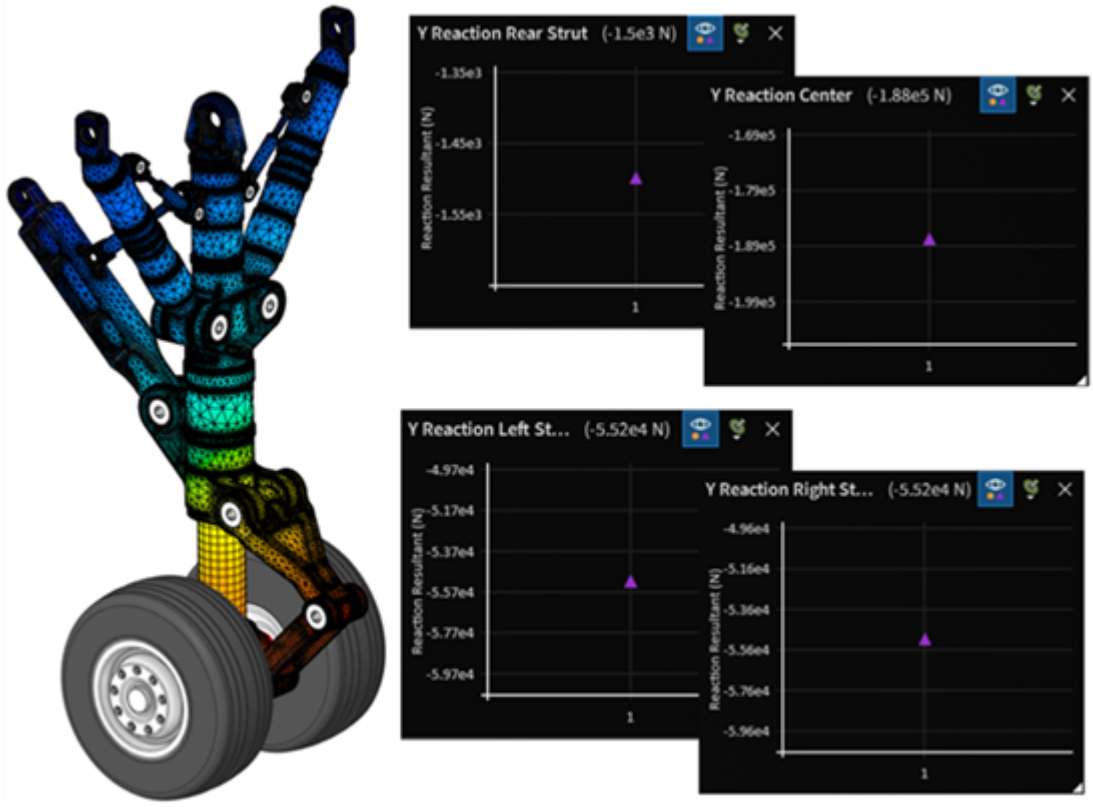
2.12.4. Reactions for Remote Displacements (Explore)

In the Explore stage, reaction resultants are now available for remote displacements, which enables you to review static equilibrium for structural models.



2.12.5. Reaction Resultants (Refine)

In the Refine stage, you can now monitor reaction resultants on individual support faces, which gives you more flexibility for defining supports and monitoring results.



2.13. Import/Export Support

- The import and export of Ansys Stride (*.stride) files is now supported.
- The following have been deprecated at Release 2025 R1:

Import Support

  - **ACIS** - *.asab, *.asat formats are not supported.
  - **Revit** - *.rvt, *.rfa files are not supported.

Export Support

ACIS

  - Options for exporting **Part manufacturing information** have been deprecated.
  - Only version V5 is supported for export.

2.14. Performance

Discovery Release 2025 R1 continues to address reported issues in application performance and modeling responsiveness. You may notice performance improvements in all stages from refresh rates to solution rendering, including:

- better performance when working with large models,
- better overall progress and messaging for modeling operations, add-ins and customization,
- improved performance for models in the Model stage,
- improved and interactive background task management, and
- better overall performance and fewer unresponsive events.

2.14.1. Large Models

Performance is improved when working with large models. You can:

- Convert geometry imported or created in Discovery to lightweight using the **Convert to Lightweight** option in the context menu from the Model tree.
- Convert lightweight geometry to heavyweight using the **Convert to Heavyweight** option in the context menu from the Model tree.
- View the lightweight state on the component parent node when all subcomponent bodies are lightweight.
- Open and save lightweight files for large models as Ansys Stride (*.stride) files.

2.14.2. Hardware Support Improvements

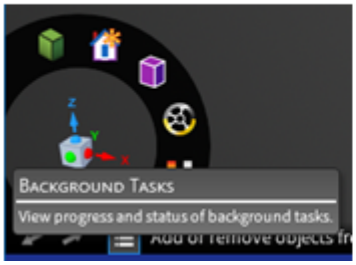
Also worth noting is Discovery's improved hardware support with the ability to run on a wider array of hardware configurations. Discovery will attempt to run with the existing hardware configuration. If resources are inadequate to solve simulations in Explore or Refine, Discovery will, at a minimum, run in the Model stage. This is particularly relevant for remote sessions.

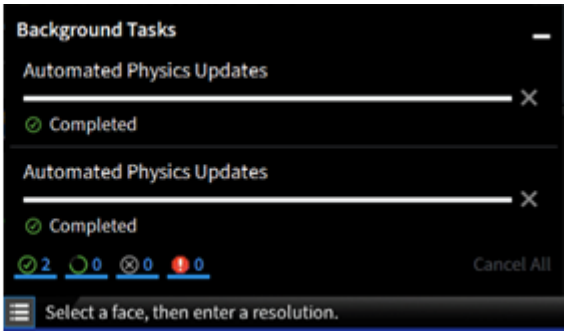
2.14.3. Responsiveness with Modeling Tools

Discovery is now more responsive when using modeling tools on complex geometries. A progress message box may now display so the current operational status is clear. Also, the SpaceClaim API progress tracker will now show a progress message box if the add-in requests it.

2.14.4. Background Tasks

Background tasks and their progress can be observed and cancelled.

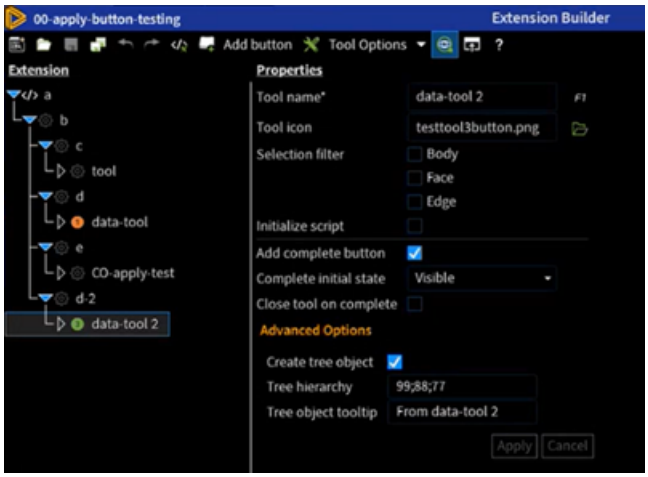




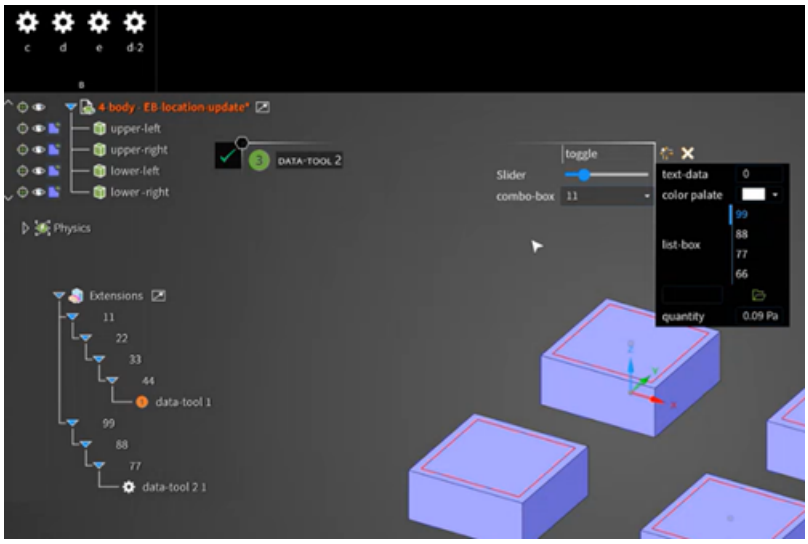
2.15. Extension Builder

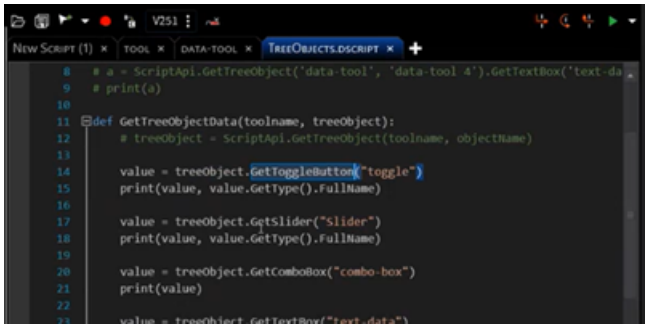
The **Extension Builder** allows you to add customizable objects to tools, which will then appear in the **Extensions tree**.

When adding a new tool, select **Advanced Option** from **Add Button > Properties Panel**. This option enables you to create a tree object, organize it into a hierarchy under the extension entry object, and add a tooltip.



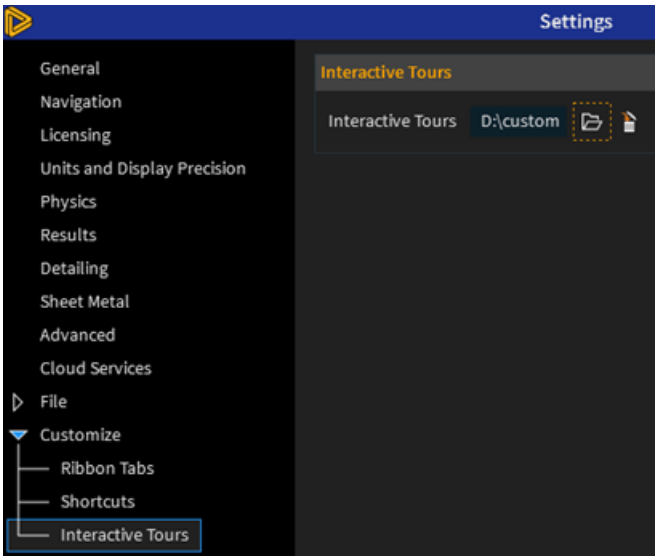
Multiple customizable objects from various tools can be added to the same tree, where they can be edited and referenced in scripts. For example, in previous versions, applying four different forces required four separate tools. Now, you can add multiple forces within the same tool.





2.16. Interactive Tours

- Links to Ansys Learning Hub have been added to the end of all the tours to provide subscribers with access to simulation courses. Those with no subscription will use a different link to access the courses on Ansys Innovation Space.
- In the **Settings** panel, under **Customize > Interactive Tours**, you can load customized interactive tours that you or your organization have created. Browse to the folder that contains the customized interactive tour resources. You can access the custom tour from the **File Menu > Home Page** under the **Custom Tours** tab.



- Discovery Learn Concepts have been added to the Static Structural interactive tour. They explain engineering concepts to help you learn the why along with the how-to. Click the embedded links in the tour to learn more about the physics used in the simulation.

2.17. Installation and Licensing

- If you install the Discovery product package, it contains a full installation of Discovery. If you install Discovery as a component of another Ansys product package, such as an Ansys Fluids or Ansys Structures package, you may now choose between a full or reduced Discovery installation. The reduced option is suitable for preprocessing only. It does not include the Fluent or MAPDL solvers, thereby limiting the use of the Refine stage as well as impacting physics transfer downstream.

- Discovery has moved to a Pro, Premium, Enterprise (PPE) licensing scheme. The former Discovery Modeling is now Discovery Pro, the former Discovery Simulation is now Discovery Enterprise, and a new mid-tier offering, Discovery Premium, is now available.

Capability Levels and Licensing Requirements	Description
Discovery Enterprise – Provides access to all geometry modeling and simulation capabilities, including advanced capabilities such as fluid-solid heat transfer, radiation, and electromagnetics physics. – Requires **disco_level1 + disco_level2 + disco_level3** licensing increments.	– Permits all modeling operations. – Permits structural, fluids, and electromagnetics solves in the Explore stage and electromagnetics and fluids solves in the Refine stage. → For structural, provides access to solving in the Refine stage but an appropriate flagship solver license is also required. → For fluids solves in the Refine stage, you can use the LiveGX solver, which is the default and does not require an
Discovery Premium – Builds on Discovery Pro by adding access to advanced modeling capabilities and additional structural	

Capability Levels and Licensing Requirements	Description
and thermal physics, as well as fluids physics. – Requires **disco_level1 + disco_level2** licensing increments.	additional flagship license. If you disable the LiveGX solver, an appropriate flagship solver license is also required. → Electromagnetics is supported in Discovery Enterprise only and does not require an additional flagship license.
Discovery Pro – Provides access to most geometry modeling capabilities, plus support for entry level structural physics. – Requires the **disco_level1** licensing increment.	– Permits most modeling operations. – Blocks creation and editing of physics objects other than entry level structural physics. – Blocks solving other than entry level structural physics. – Permits read-only access to the Explore and Refine stages for all remaining physics. – Permits read-only access to results and post-processing for all remaining physics. – Enables you to run your choice of Discovery Modeling, SpaceClaim, or DesignModeler. However, you may run only one of these products at a time.

Also see Licensing (p. 20) for information about licensing options on the Settings panel.

2.18. Ansys Discovery Technology Showcase: Example Problems

For 2025 R1, the following examples have been added:

- Planar Inverted-F Antenna Behind a Radome
- Parallel Flange Channel and Plate Bolted Connection
- Ahmed Body Drag Coefficient
- Gyroid Heat Exchanger Thermal Performance

For ease of access to the manual, a link has been added from Discovery's Home Page to the *Ansys Discovery Technology Showcase: Example Problems* manual in the Online Help.

You can download project files directly from the *Ansys Discovery Technology Showcase: Example Problems* manual.

2.19. Ansys 3D Design Verification Manual

For 2025 R1, the following test cases have been added:

- Steady-state Laminar Flow
  - Laminar Flow Between Rotating and Stationary Cylinders
- Steady-state Thermal-electric Simulation
  - Centerline Temperature of an Electrical Wire
- NAFEMS Benchmarks
  - P09 - Free Vibration Benchmarks, Abbassian, F., Dawswell, D.J., and Knowles, N.C.
    - → P09-T4 - Cantilever with Off-center Point Masses
    - → P09-T6 - Circular Ring – In-plane and Out-of-plane Vibration
    - → P09-T15 - Clamped Thin Rhombus Plate
    - → P09-T33 - Free Annular Membrane

You can download project files directly from the *Ansys 3D Design Verification Manual*.

3. Discovery Caveats and Known Issues

At the time of publication, caveats and known issues in Release 2025 R1 of Discovery include those listed below. Also refer to the *Ansys, Inc. Known Issues and Limitations* document.

- When solving a simulation that combines fluid-solid heat transfer and internal radiation with a large number of faces included in the view factor computation, the solver may become unresponsive. **Workaround**: This is a software limitation and there is no workaround. [926961]
- When using Discovery remotely, you may experience lagging when interacting with the graphical user interface. **Workaround**: Some improvement may be possible with changes to RDP settings. If possible, use Discovery locally. [1000305]
- When using remote desktop, Discovery may not connect to Workbench or launch from Workbench. **Workaround**: Disable the http_proxy on local grpc connections by setting either *no_grpc_proxy=localhost,127.0.0.1,0.0.0.0* or *no_proxy=localhost,127.0.0.1,0.0.0.0*. [1128167]

SpaceClaim Release Notes

Release 2025 R1 of SpaceClaim contains the following.

1. SpaceClaim Product Advisories

SpaceClaim users should be aware of the advisories described here.

SpaceClaim Geometry Kernel

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

Installation

SpaceClaim has been removed from the Ansys unified installers and the standalone Discovery product installation at Release 2025 R1.

Licensing

The **Legacy Named User Subscription** option is no longer supported. You should migrate to the web-based Ansys Named User licensing.

API Version Support

API version V22 has been deprecated at Release 2025 R1 and will be removed at Release 2025 R2.

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

2. SpaceClaim Changes Overview

The following changes have been made at Release 2025 R1:

2.1. Installation

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

Downloading the Installation Package From the Customer Portal

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

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

From the Command Line

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

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

Obtaining SpaceClaim as a Component of Another Product

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

2.2. Scripting

- The current Scripting version is v251.
- API version V22 has been deprecated at Release 2025 R1 and will be removed at Release 2025 R2.

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

2.3. Import/Export Support

- The legacy SpaceClaim *.scdoc files are imported using a translator. Opening these files may take longer due to the geometry conversion.
- The saving of legacy SpaceClaim *.scdoc files is no longer supported. You can only save files in the *.scdocx format.
- The import and export of Ansys Stride (*.stride) files is now supported.

- The following have been deprecated at Release 2025 R1:

  Import Support

  - **ACIS** - *.asab, *.asat formats are not supported.
  - **ICEM CFD** - *.tin files are not supported.
  - **Revit** - *.rvt, *.rfa files are not supported.

  Export Support

  - **ACIS** - Only version V5 is supported for export.
  - **CATIA V5 Assembly** - *.CATProduct export is not supported.
  - **VDA** - *.vda files are not supported.

  Part Manufacturing Information

  Options for importing/exporting **Part manufacturing information** have been deprecated for ACIS (import and export), CATIA import, CREO Parametric import, NX import, STEP import.

  Translation Technology

  Translation Technology options for AutoCAD import and export have been deprecated.

3. SpaceClaim Caveats and Known Issues

At the time of publication, caveats and known issues in Release 2025 R1 of SpaceClaim include those listed below. Also refer to the *Ansys, Inc. Known Issues and Limitations* document.

- Script indices may have changed with the switch to the Parasolid kernel and should be verified at Release 2025 R1.

DesignModeler Release Notes

1. DesignModeler Product Advisories

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

2. DesignModeler Enhancements

DesignModeler 2025 R1 has no new features or enhancements.

3. DesignModeler Caveats and Known Issues

At the time of publication, there are no caveats or known issues in Release 2025 R1 of DesignModeler. Also refer to the *Ansys, Inc. Known Issues and Limitations* document.

Ansys Structural Products

Licensing

At Release 2025 R1, Ansys Mechanical products have transitioned to the new Ansys Common Licensing model. Additional license increments have been added. The previous bundled license increments that enable multiple applications (for example, ansys, mech2, mech1) remain unchanged. For more information, see Summary of New and Changed Features in the *Ansys, Inc. License Management Guide*.

Release notes are available for the following Ansys Structural products:

Mechanical Application Release Notes

This release of the Ansys 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

Release 2025 R1 includes new features and enhancements that result in product behaviors that differ from previous releases. These behavior changes are presented below

- **Energy Contribution**. For the **Energy Contribution** result type, if you specify a single value in the **Mode** property, the graph displays as a bar chart for each associated body. Previously, the graph displayed as a single-column mosaic.
- **Mode Contribution**. Now, when you evaluate a **Mode Contribution** result, using either a **Load Step Number** or an **RPM Selection**, the application automatically evaluates the result for all load steps or RPM values. When you change the load step value or RPM of the corresponding

properties, the display of the result data changes per your selection. Previously you had to re-evaluate the result when you changed the value of these properties.

- **Fracture Object**. For the sole property of the **Fracture** object, **Re-mesh Hex-dominant to Tetrahedral**, the **On** option has been renamed **Entire Part**.

- **Fracture Controls**. Note the following changes in behavior for the **Fracture Controls** analysis settings:

  – When the **Large Deformation** property is set to **On** and you 1) then insert a **Fracture** object, or 2) the **Fracture** property of the **Fracture Controls** category of the **Analysis Settings** is set to **On** (default), the application now automatically sets the **SIFS** or **C*-Integral** properties of the **Fracture Controls** to **No**.

  – The **C*-Integral** property (fracture parameter) computation is only supported when a creep material is used and the **Creep Effects** property of the **Creep Controls** is set to **On**. This requirement was not present in the previous release.

- **RPM Selection for Loading Conditions**. For a multiple step **Harmonic Response** analysis, when you set the **Multiple Step Type** property of the **Analysis Settings** > Step Controls to **RPM**, the **RPM Selection** property now provides a drop-down list of available selections. Previously, you had to enter the desired load step value to specify the frequency.

- **Dummy mass** in **Motion**. The mass and mass moment of inertia of interface nodes and dummy bodies in flexible bodies have been reduced. This reduction minimizes their effect on the solution and improves accuracy for **Eigenvalue** analysis. In some scenarios, results in 2025 R1 will therefore differ from those obtained using 2024 R2 and previous versions.

- **Iterative Solver Type**. The ICCG solver is now limited to the **Magnetostatic** analysis only and this support may be removed entirely in a future release. See the Incomplete Cholesky Conjugate Gradient (ICCG) Solver section in the *Mechanical APDL Feature Archive* for additional information.

- **Imported Condensed Part**. Now, when you are working with an **Imported Condensed Part**, if the application reduced one or more master nodes during generation pass, those master nodes and their elements are not available on the mesh during the Use Pass. All the remaining master nodes will be available for scoping and Named Selection creation on the mesh.

- **Bushing Joint**. Now, when you set the **Formulation** property to **Bushing**:

  – The default setting for the **Element Coordinate System** property has changed from **Reference Coordinate System** to **Global Coordinate System**.

  – Only a Cartesian coordinate system is supported for the **Element Coordinate System** property.

  – The application no longer supports relative rotation between the **Reference** and **Mobile** coordinate systems.

  – The **Reference** and **Mobile** coordinate systems must be aligned with **Global Coordinate System** when the **Element Coordinate System** is not aligned with **Global Coordinate System**.

- **Checking Mesh Element Quality**. The application has been refactored to include a new error limit when checking the quality of the mesh. This better ensures that the mesh passes the

automatic element quality check. Previously, the error limit may have allowed some elements to incorrectly pass the quality check and as a result produce solution errors. You may still encounter this issue when resuming simulations (`*.mechdb`, `*.dsdb`, `*.mechdat`, etc.) from previous releases. Simply clearing and regenerating the mesh does not correct the issue. You need to improve the mesh's quality using a mesh control to properly mesh and solve your analysis.

- **Imported Data Details**. For imported loads, materials, and trace data, the **Details** pane for the associated objects has changed. The **Legend Controls** category has been renamed **Information** and now only includes the **Minimum Source** and **Maximum Source** properties that display the imported minimum and maximum data values associated with the object. You can make legend value changes using the legend displayed in the **Geometry** window.
- **Video Export using an Ansys Academic License**. If you are using an academic license for Ansys Mechanical, the application no longer supports the ability to export MP4 video files.
- **Group Similar Objects**. When grouping custom objects, objects are grouped by the object caption defined in the extension definition file. The resulting folder is also named after this caption.
- **Rename Based On Definition**: When you automatically rename a custom object based on its type and scoping, the new object name will include the caption defined in the extension definition file.

2. Documentation

The following documentation changes and/or enhancements were made for Release 2025 R1:

- **Probes**. The documentation for probe capabilities has been reorganized and consolidated in the Result Output section of the Mechanical User's Guide.
- A new real-world problem to highlight features in the Mechanical application has been added: Dynamic Simulation of a Printed Circuit Board Assembly Using Modal Analysis Methods in the *Mechanical Technology Showcase: Example Problems*.

3. Performance

The following performance enhancements were implemented for Release 2025 R1:

- During the solution process, based on model size, the application now creates the input file significantly faster. For large models, this process is as much as three times faster than the previous release.

4. Add-ons

The following updates were made to the Mechanical add-ons for Release 2025 R1:

- The **DesignLife** add-on now offers the following capabilities:
  - **Hybrid Loading Events** including time-step loading

  - **Temperature** load provider for **Time Domain Loading Events**
  - **Virtual Strain Gauge** analysis
  - The display of **Least Safety Factor** tabular data for evaluated results
  - Fatigue life estimation for MSUP Harmonic analysis with **Residual Vectors** is now possible. This newly introduced manual-selection option helps you to perform fatigue life estimation for **Harmonic** loading including a residual vector, which was not supported previously.
  - Manual selection of a Modal **rst** file for the **Modal-Transient** case in a **Time Domain** analysis.
  - **s3t** or **dat** file input for a **Time Series Load**
  - The **Solution Location** option is exposed under **Analysis Settings**, allowing you to select between AveragedNodeOnElement, WeldHotSpot, NodeOnElement or Element options
  - **Surface Nodes Only** and **Resolve to Local Analysis Settings** for crack analysis in solid elements
  - **Advanced Analysis Settings** for **Equivalent Life** calculation and **Output Stats** for reporting minimum and maximum combined stress
  - **Thermal Offset** for **Vibration Fatigue** projects with an upstream **Thermal Analysis** system
  - Material names in **Materials Assignment** now support special and Japanese characters
  - A new parametric studies option for **Certainty of Survival** can be set as a parameter design point
  - **Workflow improvements**:
    - → The DesignLife add-on now only passes FE files of selected environments, so the solution of every upstream system is no longer required
    - → Multiple DesignLife systems can now be connected and can also be intercalated with other systems.
    - → The DesignLife add-on can load environments from upstream or downstream systems
    - → A DesignLife system can be independent if only manual **rst** files are loaded.

- The **NVH Toolkit** add-on now includes support for:
  - The use of **Grouped Input and Output Remote Points** and **h5** file export from the **FRF Calculator**
  - The **FRF Calculator** also allows frequency spacing based on **Harmonic Response Analysis** and allows import of **Damping** data from an Excel file.
  - **Pre-Test Worksheet** filter mode options for **Absolute Participation Factor** and **Relative Frequency Tolerance**

  - The **MAC Calculator** now allows the use of distributed **rst** files for **File2**

- The **Rotordynamics** add-on has been added and includes the following capabilities:

  - Compatibility with every officially-supported language.

- The **Forced Response** add-on now includes support for:

  - Dark, Classic, and Light modes.

- The **Crystal Plasticity (CP) Microstructure** add-on has been added and includes the following capabilities:

  - Generates element (mesh) and materials data files that accurately capture grain geometry and texture

  - Serves as a pre-processor for crystal plasticity (CP) simulations in Mechanical APDL.

- The **Bolt Tools** add-on now offers:

  - A **Create Contacts** option added to generate contacts between the imprinted faces in the **Cone of Compression Imprint Wizard**.

- Updates for the **Additive Manufacturing add-ons** are described in the Additive Manufacturing Release Notes:

  - LPBF Process Add-on

  - Distortion Compensation Add-on

  - DED Process Add-on

  - Sintering Process Add-on

5. General

The following general enhancements were made at Release 2025 R1:

- **License Override**. An environment variable has been factored into the new release that enables you to override the default ordering of licenses the application uses and specify your own order. When you first install the application, and prior to launching it, you can use the system Environment Variable, **`MECHANICAL_LICENSE_OVERRIDE`**, to override the default ordering of licenses the application uses. The application reads this variable the first time you open the application and overwrites the system file that specifies the default license order.

- **Unit Systems**. A new unit system is now available: **Metric (mm, g, N, ms, V, A)**. For analyses that use this unit system and that have the **Solver Target** property set to **Mechanical APDL**, the application sets **nmm** for the **Solver Unit System** property (**Analysis Settings** > Analysis Data Management).

- **Performance Improvement**. Release 2025 R1 speeds up the process of loading and opening projects by optimizing how loads with tabular information are stored and sorted. Large, complex models that used to take more than 30 minutes to open now take approximately 15 seconds, a performance improvement of over 99%.

6. Graphics

The following graphical enhancements were made at Release 2025 R1:

- **Copy and Paste Geometry Window Content**. The preference (Image to Clipboard Mode) that controls how you copy and paste the content of the Geometry window has been updated. Now, when you use either the keyboard shortcut **[Ctrl]**+**[C]** or the Images menu option, **Image to Clipboard**, you can specify additional export options. The **Image to Clipboard Mode** preference (Options > Graphics > Image Export) includes the options **Current Display** (default) and **Use Image Export Settings**. When set to **Current Display**, the application copies the content of the window using the current view. When set to **Use Image Export Settings**, the application copies the content of the window based on the settings of the **Image to Clipboard Preferences** dialog. This dialog displays automatically when the **Show Preferences Dialog** preference is set to **Yes**, but only when you use the **Image to Clipboard** option of the **Images** drop-down menu.
- **Exporting Animations**. Now, when you export an animation, a new dialog provides options to adjust the width and height of the generated video.
- **Viewing Exported Animations in Web Browser**. MP4 video files exported from Mechanical can now be opened in web browsers.
- **Result Display Enhancements in Batch Mode**. In batch mode, you can now export images of results that use the Geometry display options **IsoSurfaces**, **Capped IsoSurfaces**, and **Section Planes**.

7. Geometry

The following geometry enhancements were made at Release 2025 R1:

- **Material Consolidation**. The **Model Import** feature, available when you open Mechanical independently, has a new property: **Material Consolidation**. This property enables you to choose whether to automatically merge identical materials included in the source file into a single material.

8. Contact and Connections

The following contact and connection enhancements were made at Release 2025 R1:

Contact Enhancements

- **Contact Region**. The following **Advanced** category properties of the **Contact Region** object can now be parametrized:
  - **Penetration Tolerance Factor**
  - **Penetration Tolerance Value**
  - **Normal Stiffness Factor**
  - **Normal Stiffness Value**

- **Stabilization Damping Factor**
- **Pinball Radius**

9. Mesh

For Release 2025 R1, the following mesh enhancements have been made:

- **Display Vectors of Element Normal**. A new preference is available in the Graphics category of the Options dialog that enables you to display a normal vector on each element. This display is available from the **Mesh** object and when you have the **Show Mesh** option activated.

For all other mesh related enhancements, refer to the Meshing release notes.

10. Additive Manufacturing Process Simulation

See Additive Manufacturing.

11. Acoustics Analyses

For Release 2025 R1, the following enhancements were made for Acoustics Analyses:

- **Morphing Region**. The **Morphing Region** object, available for Harmonic Acoustics analyses, has a new property: **Sliding Constraints**. This property enables you to force the nodes of selected planar faces to stay in the plane during mesh adaptation (morphing) to accurately represent the presence of symmetry or ground.
- **Mass Source**. You can now scope the **Mass Source** excitation directly to nodes or to a node-based Named Selection.

12. Analysis

Release 2025 R1 includes the following analysis enhancements:

- **Substructure Generation Analysis**. The **Substructure Generation** analysis now supports the Rotational Velocity loading condition.
- **RPM Selection**. Now, when you specify RPM as the step type during a multiple step **Harmonic Response** analysis, the **RPM Selection** property that is available for certain loading conditions and most results, provides a drop-down list of available selections. Previously, you had to entry a value manually.
- The Welding Toolbox extension has been moved to beta status.

13. Fracture

The following Fracture analysis enhancements were made at Release 2025 R1:

- **Multi Body Scoping Support for Analytical Cracks**. You can now scope all analytical crack types, except the **Semi-Elliptical Crack**, to multiple bodies for fracture parameter calculation and crack growth analysis.
- **SMART Crack Growth**. Enhancements to this feature include:
  - Support for restarting the solution from any intermediate load step.
  - Support for tabular temperature loading, including spatial varying loading.
  - **New Crack Growth Laws**:
    - → **Walker Equation**
    - → **Forman Equation**
    - → **Tabular Fatigue Law**
    - → **NASGRO Equation V3**
    - → **NASGRO Equation V4**

    In addition, when you specify either Paris' Law or the Tabular Fatigue Law, you can now include one of the following crack-closure functions: **Elber Crack-Closure**, **Schijve Crack-Closure**, **Newman Crack-Closure**, or **Polynomial Crack-Closure**.
  - **Crack Increment Options for SMART Crack Growth**. The options to simulate fatigue or static crack growth using the SMART Crack Growth feature have new crack increment properties that enable you to define minimum and maximum increment treatments and their values more specifically.
  - **Pressure Load Support for Crack Initiation**. You can now apply a normal Pressure load to the new crack faces generated during the crack growth process of the Crack Initiation feature, using the **Crack Growth** option of the **Scoping Method** property and **New Crack Faces Only** option of the **Apply To** property of the **Pressure** object.
- **Specifying Pressure and Force Loads**. Now, when your analysis includes a crack, you can specify Pressure and Force loads using the **Direct** option for the **Applied By** property.

14. Structural Optimization

The following Structural Optimization analysis enhancements were made at Release 2025 R1:

- **Optimization Type Default Setting**. The **Optimization Type** property has a new default setting: **Topology Optimization - Mixable Density**. Although not a new option, this method is an enhanced version of the previous default setting, **Topology Optimization - Density Based**.
- **Options for User Defined Criterion**. The Primary Criterion feature now:
  - Enables you to specify Equivalent Radiated Power as a Base Result for Harmonic Response analyses when you are using the **Topography Optimization** method. In association with this new option is the **Average of Decibel Level** option of the **Frequency Reduction**

property. Using this new combination of options, you can use the **Primary Criterion** object to optimize the Equivalent Radiated Power Level.

  - Includes changes to the **Spatial Reduction** property:

    → The option **Average** has been renamed **Arithmetic Average**.

    → A new option, **Continuous Average**, calculates the criteria value by averaging the product of all displacements multiplied by the element area of influence.

- **Defining Additional Exclusion Regions**. The Local Design Restriction object includes a new property: **Type**. This property enables you to specify geometry that the application does not optimize or specific faces from which the application does not move any nodes from the original location.
- **Visualizing Thermal Results**. The **Structural Optimization** analysis now supports temperature and heat flux results when you have an upstream thermal analysis.
- **HPS Support for Structural Optimization**. When running background or remote solutions, you can now specify the HPC Platform Services (HPS) application for **Structural Optimization** analyses.

15. Motion

The following enhancements were made to Motion at Release 2025 R1:

- **HPS Support for Motion**. When running background or remote solutions, you can now specify the HPC Platform Services (HPS) when using the **Motion** analyses.
- Displacement, velocity and acceleration results can be expanded on **Imported Condensed Parts** in a **Motion** analysis.
- **Condensed Parts** and **Imported Condensed Parts** now support **Beam** bodies with a **Circular** cross-section for a **Motion** analysis.
- **Condensed Part Damping** is now customizable using **Stiffness Proportional Ratio** and **Damping Ratio** solver controls.
- A **Modal Reference Position** option for **Body Properties** and a **Moving Reference Frame** option for **Marker** objects are now available to allow to improved solution convergence in **Motion** analyses.
- **High-Speed Rotation** and **Large Deflection** **Solver Controls** have been added to allow improved performance under those scenarios.
- Multiple bodies created by Motion **Tire**, **Links** and **Drivetrain** objects can now be hidden or displayed using **Hide Body** and **Hide all Other Bodies** options.
- A **Contour** display is now available for **Tie Contact** results such as Normal Force, Penetration, Friction Force, Friction Slip, and Force Magnitude.

16. Loads/Supports/Conditions

The following loads/supports/conditions enhancements were made at Release 2025 R1:

- **Apply Remote Boundary Conditions to Rigid Bodies**. Now, when you apply a Remote Force, Remote Displacement, or a Moment, to a body whose **Stiffness Behavior** property is set to **Rigid**, a new property displays: **Apply to Rigid Body Pilot Node**. Supported for body selection only. When activated, the application applies the remote boundary condition to the pilot node at the centroid of the rigid body which avoids any unwanted moments.

- **RPM Selection**. For a multiple step **Harmonic Response** analysis, when you set the **Multiple Step Type** property of the **Analysis Settings** > Step Controls to **RPM**, the **RPM Selection** property now provides a drop-down list of available selections. Previously, you had to enter the desired load step value to specify the frequency.

- **Rotational Velocity**. The Substructure Generation Analysis analysis now supports the **Rotational Velocity** loading condition.

- **Mass Source**. You can now scope the **Mass Source** excitation directly to nodes or to a node-based Named Selection.

- **Convection**. The **Convection** load now supports the following features:

  Element APDL Name

  This property enables you to assign an APDL parameter name to the surface element type number so that you can programmatically identify the convection element type and real constant set. You can then reference and use these parameters in a **Commands (APDL)** object.

  Tables for Film Coefficient and Ambient Temperature Magnitudes

  You can now use multivariable tables of data to specify the magnitudes of the film coefficient and/or the ambient temperature for a convection boundary condition.

17. Imported Loading Conditions

The following imported loading condition enhancements were made at Release 2025 R1:

- **Importing Load Data from External Data**. Using the **External Data** feature, you can now import pressure data from CFD General Notation System (`.cgns`) files generated from a Computational Fluid Dynamics (CFD) analysis. A new Imported CFD Pressure load is supported that enables you to specify the mapping settings and review the mapping results. This capability is supported for Coupled Field Harmonic, Harmonic Acoustics, and Harmonic Response analyses only.

- **Displaying Source Point Data**. Now, when you import an Imported CFD Pressure in CGNS file format or an Imported Velocity in H5DPF file format, you can display source point data information using the **Display Source Points**, **Display Source Point Ids**, and **Display Interior Points** properties.

- **Mapping Validation**. Now, when you import an Imported CFD Pressure in CGNS file format or an Imported Velocity in H5DPF file format, you can specify a Validation object with the **Type** property set to **Source Value**, to evaluate the quality of the mapping on the target mesh with respect to the original source point values.

- **Imported Remote Loads**. The **Imported Remote Loads** object has a new property: **Coordinate System**. Use this property to specify a desired Cartesian coordinate system to properly position the imported loads on your target geometry in the event there are differences in position and alignment compared to the source data.

  This object also has a new context (right-click) menu option: **Delete Children**. This option automatically deletes all child objects.

- **Specifying Upstream Named Selections for Imported Loading Conditions**. Previously only supported for **Imported Velocity**, now, for any of the imported loading conditions listed below, the **Source Bodies** property provides the option **Named Selection**. Using this option, you can select Named Selections defined in the upstream system/source file. The use of Named Selections can improve performance by only working with the source nodes specified in the Named selection.

  – **Imported Body Temperature**

  – **Imported Temperature**

  – **Imported Cut Boundary Constraint**

18. Solution

The following solution enhancements were made at Release 2025 R1:

- **Ansys Cloud Burst Compute**. A new remote solution option is now available: **Ansys Cloud Burst Compute**. This option enables you to submit and manage jobs using the **Ansys Cloud**. This capability has specific access and administration requirements.

- **Resource Prediction**: For **Harmonic Response** analyses, the **Resource Prediction** feature now supports the **Variational Technology** setting of the **Solution Method** property.

- **HPS Support for Structural Optimization and Motion**. When running background or remote solutions, you can now specify the HPC Platform Services (HPS) application for **Structural Optimization** and **Motion** analyses.

19. Explicit Dynamics

The Workbench application provides multiple 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 2025 R1 are described below.

Explicit Dynamics

For the Explicit Dynamics system, there are no enhancements for Release 2025 R1. The Explicit Dynamic Analysis System is now in maintenance mode. Please make sure to check the release notes with every release or contact your Account Manager or local support. More information may be found in the Mechanical Beta Features document.

LS-DYNA

For LS-DYNA systems, note the following enhancements for Release 2025 R1:

- **Remote Point Electrical Behavior**. Remote Point supports a new behavior named Isopotential which is useful in battery applications to allow potential coupling between faces and edges.
- **Battery Support**. There are several objects that can be defined in an LS-DYNA system that provide support for modeling batteries, including Battery Cell, Battery Thermal Abuse, Isopotential Connection, and Circuits Short Resistance.
- **ALE Viscous**. A new material model is available in Engineering Data that allows you to specify the viscosity of a given domain.
- **ALE Material Failure**. Material failure is available in the LS-DYNA system for ALE and S-ALE simulations. This approach replaces the failed material with a dummy vacuum material by creating a dummy part.
- **Contact Check**. It's now possible to use the Mechanical application Contact Tool to check the correctness of your LS-DYNA simulation's contact specifications. You can also evaluate various contact results to see if the behavior is as expected.
- **Delamination Support**. The Mechanical application's Interface Delamination object is supported in the LS-DYNA system. You can trigger delamination of 2 layers of a composite model through a defined energy level.
- **Contact Debonding**. The Mechanical application's Contact Debonding object is now supported by the LS-DYNA system, allowing you to break a contact when a certain energy level is achieved.
- **Alternate Method to Define Composites**. The LS-DYNA system allows you to use the part composite approach instead of the element shell approach that was previously used. Composites can now be defined over a broad area.
- **Body Temperature Load Transfer**. It is now possible to transfer body temperature loads from a Steady State Thermal or Transient Thermal calculation to an LS-DYNA system using dissimilar meshes.

20. Results

The following result enhancements were made at Release 2025 R1:

- **Energy Contribution**. For the **Energy Contribution** result type:
  - You can now scope this result to geometry or elements as well as geometry- and element-based Named Selections.

  - You can now specify an energy type for this result. Options include **Total Energy**, **Kinetic Energy**, and **Strain Energy**. The application evaluates all energy types during the solution so you can switch between options without needing to reevaluate the result.

  - If you specify a single value in the **Mode** property, the graph displays as a bar chart for each associated body.

- **Mode Contribution**. Now, when you evaluate a **Mode Contribution** result, using either a **Load Step Number** or an **RPM Selection**, the application automatically evaluates the result for all load steps or RPM values. When you change the load step value or RPM of the corresponding properties, the display of the result data changes per your selection. Previously you had to re-evaluate the result when you changed the value of these properties.

- **Graph and Tabular Data for Mosaic Charts**. Now, for all results that display mosaic charts:

  - The Chart and Tabular Data category of the Details pane provides two new properties: **Legend Minimum** and **Legend Maximum**. By default, these properties display the evaluated minimum and maximum values for the result and also enable you to enter a desired minimum and maximum value within the result range to modify the graph display. If you specify a value outside of this range, the application automatically restores the default values.

  - The **Show Text on Mosaic** property of the Chart and Tabular Data category of the Details pane has a new option: **Yes (Show Refined Text)**. For this option, the chart display is refined and text only displays in cells that are representative of the distributed values, such as minimum, maximum, and values across the range.

  - Selecting a cell on the mosaic chart highlights the corresponding value (row) in the **Tabular Data** window.

- **Line Chart**. Enhancements for this feature include:

  - Support for all result-based options available under the **Solution** object of an LS-DYNA analysis, which specifically now includes results (deformation, stress, etc.), result trackers, and probes.

  - **Tabular Data** window updates include the display of units of measure for the datasets (also a new display in the chart) as well as new context (right-click) menu options that enable you show, hide, or delete your desired datasets.

  - Support for unit system conversion for the displayed datasets.

- **Frequency Response**. The **Frequency Response** result has a new Chart and Tabular Data property: **Display Mode**. This property enables you to filter graph and tabular data by mode for a specified frequency value. The application plots graph data in a bar chart based on the result type, deformation, velocity, or acceleration, versus Mode.

- **RPM Selection**. For a multiple step **Harmonic Response** analysis, the **RPM Selection** property is now available for results in the **Scope** category of the Details pane when you set the **Multiple Step Type** property of the **Analysis Settings** > Step Controls to **RPM**. Use this property to scope the result to an available RPM value from the drop-down menu. The application uses the value of the first load step as the default.

- **Mesh-based Scoping**. The following result types now support mesh-based scoping:
  - **Waterfall Diagrams**:
    - → The **Velocity Waterfall Diagram** and **Acceleration Waterfall Diagram** results support Node, Element and Element Face scoping.
    - → The **ERP Waterfall Diagram** and **ERP Level Waterfall Diagram** results support Element Face scoping when the On Demand Expansion property (**Analysis Settings** > Options) is set to **Yes**.
  - **Frequency Response**:
    - → The **Stress** and **Strain** results support Element and Element Face scoping when the On Demand Expansion property (**Analysis Settings** > Options) is set to **Yes**.
    - → The **Equivalent Radiated Power (ERP)** and **Equivalent Radiated Power Level (ERPL)** results support Element Face scoping when the On Demand Expansion property (**Analysis Settings** > Options) is set to **Yes**.
  - **Phase Response**: The **Stress** and **Strain** results support Element and Element Face scoping when the On Demand Expansion property (**Analysis Settings** > Options) is set to **Yes**.
- **Response PSD Probe**. You can now scope this probe to a node.

Mechanical APDL Release Notes

Release 2025 R1 of the Mechanical APDL™ application offers most of the capabilities from prior releases plus many new features and enhancements. Areas where you will find updates and new capabilities include the following:

For more information about this release, see the Update Guide (p. 65) section of this document and the Global Release Notes.

1. Structural

Release 2025 R1 includes new features and enhancements for the following structural analysis disciplines:

1.1. Contact

Release 2025 R1 includes the following enhancements for analyses involving contact:

1.1.1. Enhanced Double-Sided Target Surfaces

Prior to this release, the double-sided target surface option (KEYOPT(8) = 1 of the TARGE170 element) is only valid only for the Gauss detection (KEYOPT(4) = 0 of the CONTA174 element) and "Normal from contact nodes" (KEYOTP(4) = 1,2 of CONTA174 element)). Now, the double-sided target surface option is also valid for "Projection-based method" (KEYOTP(4) = 3,4 of CONTA174 element and "Unified approach" (KEYOPT(4) = 5 of CONTA174).

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

For more information, see see Double-Sided Target Surfaces in the Contact Technology Guide.

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 solution accuracy. In this release, the constraint type detection logic (KEYOPT(5) = 2 of target element) has been revised so that larger percentages of assembly models can be solved with improved robustness and accuracy, in particular for mixed constraint types (solid-to-solid, shell-solid, shell-to-shell) defined within a contact pair. For more information, see "TARGE170 Input Data" in the Element Reference.

1.1.3. Penalty And Augmented Lagrange Multiplier Introduced to Surface-based Constraint

To help resolve overconstraint conditions, penalty (KEYOPT(2) = 1) and augmented Lagrange multiplier (KEYOPT(2)=0) formulas are introduced to rigid and force-distributed types of surface-based constraint. They are suitable to relax constraint on surfaces nodes. These two formulae are also available to build spot weld connections using 3D node-to-surface contact element CONTACT175.

1.1.4. Non-linear History File Now Includes Physical Time

Physical time (for scaled wear analysis) is added into the non-linear history file, for fretting fatigue and bolt self-loosening problems where a map between simulation and physical time is needed. The total mechanical (EPTO) and total strain (EPTT) are also added in the solution tracking variable list.

1.1.5. Enhancements to Contact Data Tracking File

The contact data tracking file (CND) format is changed for easy contact diagnosis. The convergence status and maximum penetration element (contact/target side) are added, and column values are rearranged with acronym on the top. The new CND format is easy to export into other plot forms.

1.1.6. Contact Detection Improvements

Contact detection robustness has been improved by adding further checks for potential spurious contacts in cases when the default pinball may be too large. This is expected to improve both accuracy and robustness of non-linear contact problems.

1.2. Elements and Nonlinear Technology

Release 2025 R1 includes the following enhancements to elements and nonlinear technology used in structural analyses:

1.2.1. Gasket Element Enhancements

The INTER194, INTER195 3D gasket elements with thin solid option (KEYOPT(2) = 2) now support nonlinear material behavior based on ductile damage, in addition to von Mises plasticity.

1.2.2. Temperature Loading for Structural Reinforcing Elements

For structural analysis, reinforcing elements (REINF263, REINF264, REINF265) allow temperature input using both mesh-independent and standard methods. Design changes were made to elements (MESH200, REINF263, REINF264, and REINF265) and loading commands (**BFE**, **BF**, and **BFPORT**) to support this feature.

1.2.3. Follower Load Element Enhancement

The FOLLW201 element with KEYOPT(1) = 2 is now supported to specify load properties with the **SFCONTROL** command.

1.3. Material and Fracture Modeling

Release 2025 R1 includes the following enhancements to material modeling and fracture analysis technology used in structural analyses:

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. Material Curve-Fitting Enhancements

Material curve-fitting now supports shape memory alloy material models. Shape memory effect and super elasticity options are supported with uniaxial and biaxial test data.

ViscoElasticity fitting can involve multiple sets of experimental data at different temperatures for Shear and Modulus Bulk Decay. Partial solver options are now supported to address this situation with a multistep solution approach. The **TBFT** command can be issued to initiate the solves as

needed (shear terms, bulk terms, and/or all experimental data). See Partial Solve Options Example for Prony Series Models for an example.

In material curve-fitting, AI-based automatic initialization (**TBFT**,AINI) of creep material parameters is now supported. For more information, see Creep Material Models and Initialize the Coefficients.

1.3.2. New UserMat Example

An example has been added to the *Programmer's Reference* to provide information on how to set up a UserMat API in C or C++.

1.3.3. Ductile Damage Enhancement

You can now use the ductile damage model with INTER194 and INTER195 3D gasket elements with thin-solid option (KEYOPT(2) = 2: through-the-thickness, in-plane membrane, and transverse-shear deformation).

1.3.4. Fracture Parameter J-integral Calculation Enhancements

Problems with Large Deflections

The fracture parameter J-integral calculation can now provide accurate results for both 2-D problems (PLANE182 and PLANE183) and 3-D problems (SOLID185 and SOLID186) with large deflections.

Hyperelastic Materials

The fracture parameter J-integral calculation now supports isotropic hyperelastic materials with large deformation.

1.3.5. Support for Initial Mesh-Based Tabular Data Support

SMART crack-growth simulation now supports tabular loads, including those where ELEM and/or NODE are primary variables for temperature, displacement, and pressure loadings. You can now update the load tables after a load step or restart command, but you must use the initial mesh-based node and element numbering and not the current mesh after any changes (when ELEM and/or NODE are used as primary variables). For additional information and limitations, see SMART Crack-Growth Assumptions and Limitations in the Fracture Analysis Guide.

1.4. Linear Dynamics

Release 2025 R1 includes the following enhancements for structural analyses involving linear dynamics:

1.4.1. Harmonic Balance Method (HBM) Multithreading

Multithreading can be activated (using the `-nt` command line option) for HBM analysis to enable Shared-Memory Parallel (SMP) processing for core harmonic computations. This option can significantly improve the computation time for models with a higher number of linear degrees of freedom and/or harmonics. For more details, see HBM Analysis and Parallel Processing in the *Harmonic Balance Method Analysis Guide*.

1.4.2. Calculating the Element Results in a Component Mode Synthesis Generation Pass

Linear perturbation analysis is now supported for a component mode synthesis generation pass where the element results calculation is activated (*Elcalc* = YES on the **CMSOPT** command).

1.4.3. Specify Initial Displacement and Velocity Conditions

Mode superposition transient analysis now supports the specification of displacement and velocity initial conditions. This can provide more accurate results for the cases with non-zero initial displacement and velocity values.

2. Multiphysics

Release 2025 R1 includes the following enhancements for analyses involving multiphysics environments:

2.1. Acoustics

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

2.1.1. Porolastic Acoustics for Axisymmetric Element

For poroelastic acoustics, Biot's theory-based mixed displacement-pressure finite element harmonic solver is now available for axisymmetric elements, including FLUID244. For details, see Poroelastic Acoustics in the *Theory Reference*.

2.2. Thermal

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

2.2.1. Membrane Option for SHELL294

Use the new KEYOPT(1) = 1 to specify the membrane option for SHELL294. This option reduces the computational load by assuming no temperature variation through the thickness of the shell. It can be useful when the in-plane temperature variation is dominant.

2.2.2. Option for Nonlinear Convection Film Coefficient on **SOLID279**, **SOLID291**, and **PLANE293**

Use the new KEYOPT(2) to specify a nonlinear temperature-dependent convection film coefficient for current generation thermal elements SOLID279, SOLID291, and PLANE293.

2.2.3. Kinematic-based View Factor Updating

Now, in addition to frequency-based view factor updating at the substep level, you can specify kinematic-based view factor updating to only update when geometric changes in the radiation surfaces exceed prescribed thresholds using the **VFUP** command. This can be a more efficient updating strategy than frequency-based criteria which update view factors whether or not deformation or kinematic motion has significantly changed the radiation facet geometries. It is particularly useful when you have no knowledge about body deformation prior to the simulation.

2.2.4. Nonlinear Stabilization for Simulations that use the Radiosity Solver Method

A new nonlinear stabilization option is available through the *STABILIZATION* argument on the **RADOPT** command. Setting *STABILIZATION* equal to one can help radiosity solutions converge faster by allowing the film coefficient and bulk temperatures to vary with temperature during the Newton-Raphson loop. Setting *STABILIZATION* equal to two gives additional stabilization for problems with temperature-dependent emissivity.

2.3. Coupled-Field

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

2.3.1. Option for Nonlinear Convection Film Coefficient on Coupled Field Elements **PLANE222**, **PLANE223**, **SOLID225**, **SOLID226**, and **SOLID227**

Use the new KEYOPT(14) to specify a nonlinear temperature-dependent convection film coefficient for the coupled-field elements PLANE222, PLANE223, SOLID225, SOLID226, and SOLID227.

2.3.2. Enhanced Strain Formulation supported for PLANE222 and SOLID225 for Structural-Thermal-Electric-Diffusion Analysis

You can now enable the enhanced strain or the simplified enhanced formulation in a structural-thermal-electric-diffusion analysis 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.

2.3.3. User-programmable Coupled-field Analyses

User-programmable features (UPFs) are available for coupled-field elements PLANE222, PLANE223, SOLID225, SOLID226, and SOLID227 by setting KEYOPT(12) = 1. Coupled-field UPFs enable you to customize element characteristics, material models, loads, constitutive equations, and output items. The coupled-field UPFs are integrated into the element standard workflow, so you can customize element properties of interest without writing a user element or material model from scratch. Several utility functions are available to help you access element results and other element information, making it easier to program nonlinearities. See Customizing a Coupled-Field Element in the *Programmer's Reference* for details.

3. Solvers

Release 2025 R1 includes the following improvements to the solution process:

3.1. Distributed-Memory Parallel Processing Enhancements

Enhancements include:

- The default support for message passing interface (MPI) software has been upgraded to Intel MPI 2021.13.0 for both Linux and Windows. Previously, the default was Intel MPI 2021.11.0.
- When using AMD processors, the default support for message passing interface (MPI) software is now Open MPI 4.0.5.

For details and other optional supported MPI software versions, see MPI Software in the *Parallel Processing Guide*.

3.2. GPU Acceleration Feature Updates

Updates include:

- The ROCm libraries have been updated to 6.2.0. Prior releases used version 5.7.0.

- The CUDA libraries have been updated to 12.6. Prior releases used version 12.0.
- The performance of the PCG solver has been improved when running with GPU acceleration as more equation solver calculations are offloaded on the GPU card(s).

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

3.3. Updated List of Recommended Devices for the GPU Accelerator

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

- AMD Instinct MI325X
- AMD Instinct MI250X
- NVIDIA B100
- NVIDIA A800
- AMD Radeon Pro W7900
- AMD Radeon Pro W7800
- NVIDIA RTX 5880 Ada
- NVIDIA RTX A4500 Ada

The following were removed:

- NVIDIA Quadro GV100
- NVIDIA Tesla Series V100

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

3.4. New Mixed Equation Solver

A new solver called the Mixed equation solver is now available. It combines the best of both the SPARSE and PCG capabilities, improving speed and reducing memory usage. For more information, see **EQSLV** and Solution in the *Basic Analysis Guide*.

3.5. PCG Solver Enhancements

- For the PCG solver, the level of difficulty (`Lev_Diff`) has been changed to optimize their values. For more information, see Change in Level of Difficulty Values for the PCGOPT Command (p. 66) and **PCGOPT**.
- The PCG solver requires less memory than in previous releases to solve models when:

  - Running on higher core counts (more than 64 cores).
  - Constraint equations and/or coupling equations exist in the model.
- The PCG solver performance has been improved for certain models containing many contact pairs that were split using the contact splitting feature (**CNCHECK**,SPLIT).

4. Other Enhancements

The following other enhancements have been added:

4.1. Access to Python, PyMAPDL, and PyDPF from within Mechanical APDL

You can now use the Python language, as well as PyMAPDL and PyDpf, directly within Mechanical APDL without any additional installation steps required. Use the new ***PYTHON** / ***ENDPY** process control commands to begin and end a block of content that is interpreted as Python commands. The ability to write Python blocks directly in a Mechanical APDL input command listing provides access to built-in Python features as well as popular third-party libraries like NumPy. To include libraries that are not available by default, you can configure Mechanical APDL to use a custom Python environment. See the ***PYTHON** command definition for details.

4.2. CAD Support Updates

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

- Added support for:
  - NX 2406 (limited to Windows 10 and 11 Professional and Enterprise versions)
  - Creo Parametric 11.0 (limited to Windows 10 and Windows 11 Version 21H2 or higher)
- Removed support for:
  - NX 2312
    - → Red Hat Enterprise Linux 8
    - → SuSE Linux Enterprise Server and Desktop 15
  - NX 2212 (on all operating systems)
  - Creo Parametric 8.0 (on all operating systems)

For more information, see Introduction to the Connection Functionality in the *Connection User's Guide*.

4.3. Two New Coded Database File Commands

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

- **DBLOCK** – Defines a block of degree-of-freedom constraints at nodes.
- **FBLOCK** – Defines a block of nodal force loads.

For more information on CDB file commands, see Coded Database File Commands in the *Programmer's Reference*.

4.4. Visual Studio Compiler Update on Windows Platforms

The supported Visual Studio compiler version has been updated to Microsoft Visual Studio Professional 2022 Version 17.6.x. In previous releases, it was Visual Studio Professional 2019 Version 16.0.22. For more information, see Compiler Requirements for All Windows Versions in the *Ansys, Inc. Installation Guides* and Compiling and Linking UPFs on Windows Operating Systems in the *Programmer's Reference*.

5. Commands

This section describes changes to commands at Release 2025 R1:

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:

- ***ENDPY** – Terminates a block of content that is interpreted as Python commands.
- ***PYTHON** – Initiates a block of content that is interpreted as Python commands. Use the APDL process control commands, ***PYTHON** and ***ENDPY**, to add Python code directly in a Mechanical APDL input listing.

5.2. Modified Commands

The following commands have been enhanced or otherwise modified:

- **BF** – Defines a nodal body-force load. The command now supports temperature body-force loads on MESH200 elements using the mesh-independent method.
- **BFE** – Defines an element body-force load. The command now supports temperature body-force loads on reinforcing and MESH200 elements.
- **BFINT** – Activates the body force interpolation operation. Use the new *Fmat* argument to define the output file format (default is the blocked format).

- **BFPORT** – Transfers body-force loads (heat generation, temperature, and body-force density) from selected MESH200 elements to reinforcing elements. The command now supports temperature body-force loads on reinforcing elements.
- CMBLOCK – The argument *KOPT* now supports SMART crack-growth analysis with *Entity* = ELEM and NODE and it is limited to solid element components and surface node components.
- **LANBOPTION** – Specifies Block Lanczos eigensolver options. Use the new *Memory_Option*, MININCORE to drastically minimize memory usage when your hardware configuration has limited memory resources.
- **OSRESULT** – Controls the selected result data written to the database, helping to reduce the size of the results (RST) file. The results output has been enhanced to include more quantities, such as Biot's effective stresses, energy densities, microplane damage, microplane homogenized damage quantities, thermal flux and gradient components (CPT elements), porous media free strain, anisotropic damage quantities, and structural dissipation rate.
- **RADOPT** – Specifies Radiosity Solver options. Use the new *STABILIZATION* argument to enable stabilization terms of the nonlinear temperature-dependent convection film coefficient and improve convergence behavior.
- **SFCONTROL** – Defines structural surface-load properties on selected elements and nodes for subsequent loading commands. Use the new short input format to enter a single label (surface-load property) and value.
- **TBFT** – Performs material curve-fitting operations. The material models for curve-fitting now include shape memory alloy. The **TBFT**,SET command now supports partial solver options to set the reference temperature with *Option4* = COMP and *Option5* = PSHEA (to solve only for Shear Modulus parameters) or PBULK (to solve only for Bulk Modulus parameters).
- **VFUP** – Controls view factor updating at the substep level for a coupled-field radiation analysis that includes large-deflection effects. Use the new options, KNEX, KNIM, and KNI1, to specify kinematic-based view factor updating criteria to update only if geometric changes to radiation surfaces surpass specified thresholds.

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 2025 R1:

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

6.1. New Elements

There are no new elements this release.

6.2. Modified Elements

The following elements have been enhanced:

- SOLID279, SOLID291, and PLANE293 – KEYOPTION(2) has been added to the current technology thermal elements to enable a nonlinear convection film coefficient.
- SHELL294 – KEYOPTION (1) has been added to specify the membrane option.
- PLANE222, PLANE223, SOLID225, SOLID226, and SOLID227 – Set the new KEYOPTION (12) = 1 to activate user-programmable features in a coupled-field analysis.
- PLANE222, PLANE223, SOLID225, SOLID226, and SOLID227 – KEYOPTION(14) has been added to the coupled-field elements to enable a nonlinear convection film coefficient.
- INTER194 and INTER195 – The thin solid option with KEYOPT(2) = 2 (membrane and transverse shear stiffness) now supports nonlinear material behavior based on ductile damage, in addition to von Mises plasticity.
- FOLLW201 – KEYOPT(1) = 2 has been added to specify load properties with the **SFCONTROL** command.
- MESH200 – This element now supports temperature element loading (**BF** and **BFE** commands) when using the mesh-independent method to define reinforcing.
- REINF263, REINF264, and REINF265 – These elements now support temperature element loading (**BFE**) when using the standard method for structural reinforcing analysis. For the mesh-independent method, apply the temperatures to MESH200 elements.

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.

7.1.1. New UserMat Example

An example has been added to the *Programmer's Reference* to provide information on how to set up a UserMat API in C or C++.

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. 68) in the Update Guide section of this document..

8. Technology Showcase: Example Problems

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

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

The following example problems have been added at this release:

- TD-71 – Effect of Fretting Wear on Non-rotational Bolt Self-Loosening.

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

9. Mechanical APDL Release 2025 R1 Update Guide

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

The following topics offer supplemental 2025 R1 product-update information presented by the Mechanical APDL development and testing teams:

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

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

9.1. Backward Compatibility

Mechanical APDL Release 2025 R1 can read database files from all prior Mechanical APDL releases. Due to ongoing product improvements and defect resolutions, however, results obtained from old databases running in new releases may differ somewhat from those obtained previously.

9.2. Feature Updates Causing Result or Behavior Changes

The following Mechanical APDL feature updates in Release 2025 R1 are known to produce program behaviors or analysis results that differ from those of the prior release:

9.2.1. Change in Level of Difficulty Values for the **PCGOPT** Command

The level of difficulty argument for the **PCGOPT** command (**PCGOPT**,*Lev_Diff*) values have been changed to more granular options. Therefore, the *Lev_Diff* values from former releases now have new meanings, as mapped in the table below. You must ensure your *Lev_Diff* values correspond to their new meanings.

Previous *Lev_Diff* Behavior	**Current *Lev_Diff* Behavior**	**Current *Lev_Diff* Values Compared to Previous Behavior Values**
AUTO — Automatically chooses *Lev_Diff* value (1 through 4)	AUTO — Automatically chooses *Lev_Diff* value (1 through 5)	
1	1	Smaller than previous level 2
2	2	Smaller than previous level 2
3	3	Previous level 2
4	4	Close to previous level 3
5	5	Close to previous level 4

9.2.2. Change in Supported Visual Studio Compiler for CUDA Libaries

The CUDA libraries have been updated to version 12.6, which only supports Visual Studio 2022 Professional. Therefore, the version used in prior releases (Visual Studio 2019 Professional) is no longer supported.

9.2.3. Default MPI Software for AMD Processors

When using AMD processors, the default message passing interface (MPI) software is now Open MPI 4.0.5. In previous releases, the default was Intel MPI.

9.3. Known Incompatibilities

No incompatibilities were introduced at this release.

9.4. Known Issues

The following issues are known to exist at Release 2025 R1:

- **Models running on AMD GPU devices while using the PCG solver may crash** — Crashes due to missing ROCm packages may occur when running on AMD GPU devices while using the PCG solver. The error message contains some or all of the following:

```
Memory access fault by GPU node-2 (Agent handle: 0x18c9150) on address 0x14f319185000. Reason: Page not preser
GPU core dump failed
forrtl: error (76): Abort trap signal
Image              PC                Routine            Line        Source
libc.so.6          000014F4E7C54DB0  Unknown               Unknown  Unknown
libc.so.6          000014F4E7CA154C  Unknown               Unknown  Unknown
libc.so.6          000014F4E7C54D06  raise                 Unknown  Unknown
libc.so.6          000014F4E7C287F3  abort                 Unknown  Unknown
libhsa-runtime64.  000014F4E5475327  Unknown               Unknown  Unknown
libhsa-runtime64.  000014F4E5474434  Unknown               Unknown  Unknown
libhsa-runtime64.  000014F4E5427677  Unknown               Unknown  Unknown
libc.so.6          000014F4E7C9F802  Unknown               Unknown  Unknown
libc.so.6          000014F4E7C3F450  Unknown               Unknown  Unknown
```

  Workaround: A potential fix to this issue is installing the latest complete ROCm package from AMD.

- **Open MPI with AMD machines limitation** — When using Open MPI, running Mechanical APDL in a nested manner is not supported. Example:

```
/syp, /ansys_inc/v251/ansys/bin/mapdl -j sample_test
```

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

9.5. Resolved Issues and Defects

The following notable issues and defects for Mechanical APDL have been resolved at Release 2025 R1:

ID	Resolution Description
1100522	The licenses for **SEXP** and **SEEXP** are now correct. **SEXP** is available to all license levels. **SEEXP** is available for the Enterprise license.
1022147	BINLIB not supporting Unicode (non-ASCII) characters has been documented. For more information, see Accessing Binary Data Files in the *Programmer's Reference*.

ID	Resolution Description
1023052	The instability when using the Open MPI library and clusters containing AMD architecture with a high core count has been documented with a workaround. For more information, see Troubleshooting in the *Parallel Processing Guide*.

9.6. Features Removed or Archived

If a legacy Mechanical APDL feature has been removed at Release 2025 R1, this section provides information about the Mechanical APDL replacement feature, its functional equivalent in another Ansys product, or another workaround.

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

The following topics are available:

9.6.1. JCG, ICCG, and QMR Solvers that do not Support DMP

The ICCG and QMR solvers have been archived, along with parts of the JCG solver that do not support DMP. For more information, see Iterative Solvers in the *Feature Archive*.

9.6.2. PLANE13

The coupled-field element PLANE13 has been archived. Although this archived element is available for use in your analysis, Ansys, Inc. recommends using its current-technology counterparts instead: PLANE222 or PLANE223.

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

There are no enhancements for Release 2025 R1.

Aqwa Release Notes

This release of the Aqwa related products contains all capabilities from previous releases. The following enhancements are available in release 2025 R1. Refer to the product-specific documentation for full details of the new features.

1. Aqwa Solver Modules

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

- **More accurate calculation of radiation force/pressure**. The impulse functions are calculated at the analysis time step interval when this interval is less than 0.1 seconds.
- **Cable pretension definition**. You can define the pretension for linear elastic cables with or without pulleys, steel wire, nonlinear polynomial line, fixed fenders, and nonlinear catenary cables.
- **Intermediate buoy or clump weight along dynamic cable disconnection**. You can define the breaking times of intermediate buoy or clump weights along dynamic cables in the time domain hydrodynamic response analysis.
- **Hydrodynamic database in LS-DYNA MCOL format**. You can output the hydrodynamic database in LS-DYNA MCOL format for further LS-DYNA MCOL ship collision or grounding analysis.

2. Aqwa Graphical Supervisor (AGS)

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

- **Animation of dynamic cable intermediate buoy or clump weight disconnection**. You can animate the procedure of buoy or clump weight disconnection. The broken buoy or clump weights will disappear from the Model Visualisation window after their breaking times during the animation of the moored structure motions of the time domain analysis results.

3. Hydrodynamic Analysis Systems

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

- **Co-simulation Settings step in the Aqwa Co-simulation Setup Wizard**. In the Aqwa Co-simulation Setup Wizard, you can define the co-simulation analysis and output time, and specify Aqwa pressure output settings. This allows for more flexible control over the output results and helps reduce the size of the result files.

- **Pressure Results Always Available**. You can now add Pressures and Motions or Internal Tank Pressures to the Hydrodynamic Diffraction Solution when the **Generate Wave Grid Pressures** option is turned off.
- **Hydrodynamic Database Reuse**. The Aqwa Workbench editor will reuse the existing hydrodynamic database as much as possible to reduce the computational cost of re-solving the Hydrodynamic Diffraction analysis.
- **Initial velocity definition**. You can now add the initial velocity of the structure COG in the Aqwa Workbench editor for the time domain Hydrodynamic Response analysis.
- **Stiffness matrix due to mooring system**. You can import an MFK file for the Connection Stiffness definition when solving the Hydrodynamic Diffraction analysis.
- **Dynamic point**. You can create a *dynamic point* in the Aqwa Workbench editor, which can be selected as an Attachment Point for any Connections but it is not associated with any Part. The dynamic point can be used for modeling bridle moorings etc.
- **Multi-structure load mapping**. You can map the hydrodynamic pressures and the accelerations at the structures' centers of gravity of an Aqwa multiple-structure model onto the Static Mechanical model through the Hydrodynamic Pressure add-on.

Ansys Composite PrepPost (ACP) Release Notes

This release of the Ansys Composite PrepPost (ACP) product contains all capabilities from previous releases. The following sections present the new features, enhancements, and known limitations for ACP in Release 2025 R1.

1. New Features and Enhancements

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

1.1. Retirement of ACP (Post)

ACP (Post) is no longer supported for any Ansys products. Ansys recommends using the Mechanical product to postprocess composite structures. The following features in the Mechanical application provide the previous core features of ACP (Post): Composite Failure Tool, Composite Sampling Point Tool, and Envelope Solution. For more information on composites postprocessing, see Composite Postprocessing in the *Mechanical User's Guide*.

A Python module is available for scripting and customizing composite postprocessing.

All ACP (Post) computations are blocked. You can still restore Ansys Workbench projects that contain one or more ACP (Post) systems. You can also open ACP (Post) to migrate the failure definitions to the Mechanical application. For information on how to migrate old projects, see ACP (Post) to Mechanical Software Migration in the *ACP User's Guide*.

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

1.2. Postprocessing in the Mechanical Application and with PyDPF Composites

There are several enhancements for composites postprocessing:

- The postprocessing of Imported Solid Models is now fully supported in the Mechanical application and through .
- The projection of the critical failure values and modes onto the skin is now also implemented for Imported Solid Models. This makes the critical failure values visible outside of the box. You can enable this feature in the Mechanical application using the **Show On Reference Surface** option.
- The Envelope Solution now supports text plots that contain information about the critical load case and failure mode.
- The processing of large `.rst` files is significantly faster (twice as fast) in this release compared to prior releases. This enhancement improves the postprocessing performance in the Mechanical application and through PyDPF Composites.
- You can postprocess cyclic symmetry models with PyDPF Composites. For more information on PyDPF Composites, see .
- The material ply types are now accessible in PyDPF Composites.
- PyDPF Composites contains new examples in its documentation and has various bug fixes.

1.3. Improvements Towards LS-DYNA

The Workbench LS-DYNA module now supports the interface layers of ACP solid models to model delamination of plies. The interface layers are converted to zero-volume LS-DYNA cohesive elements (ELFORM=19) with a cohesive material model (*MAT_COHESIVE_MIXED_MODE). In the **Engineering Data**, it is defined by material model **Bilinear for Interface Delamination**.

Also, contact debonding between bodies is now supported as well. The bonded contacts are converted to LS-DYNA tiebreak contacts with a fracture model based on *MAT_COHESIVE_MIXED_MODE. In the **Engineering Data,** the supported cohesive zone material model is **Fracture-Energies based Debonding**.

For more details, see LS-DYNA User's Guide and Interface Layer Properties in the *ACP User's Guide*.

1.4. Serialization of CAD Geometries

Imported geometries in ACP are now fully serialized which significantly improves the performance of the open and reload step in ACP. This enhancement resolves delays when opening ACP models with large geometries that have many bodies or surfaces.

1.5. Nodal Thicknesses from Lookup Tables

Starting from Release 2025 R1, nodal thicknesses are now supported for modeling plies with a thickness definition (Modeling Ply Properties) through lookup tables (Look-Up Table). The nodal thicknesses

are only computed if the option **Use Nodal Thicknesses** is enabled (Model Properties - General). Then, that additional information is used for the generation of a 3D mesh (Solid Model).

The nodal thicknesses enhancement is significant since the default thickness computation (at the element centers) may lead to an inaccurate representation of the ply thickness in certain areas, especially if the thickness becomes 0.

1.6. Support of CDB File Format

PyACP and the standalone version of ACP enable you to operate directly on APDL models via the `.cdb` file format. The updates to the `.cdb` interface of PyACP and ACP support the latest enhancements, including the **ETBLOCK** command.

1.7. Scene Lighting Modes

The ACP scene now supports three lighting modes: dark (default), light (reduced shadows), and no shadows (all surfaces fully illuminated). The light and no shadows modes improve visibility in closed structures or regions in the shade. The no shadows mode accurately represents colors in plots, but does not convey curvature information. You can switch between lighting modes using the following button in the Scene's toolbar:  .

1.8. ACCS: Cure Simulation and RTM Solver

ACCS now comes with an RTM solver that enables the simulation of infusion processes and layered composites. Results such as infusion time, fill (flow front), pressure, and velocity can be used to investigate the infusion process. The RTM solver is part of the standard ACCS license and can be easily configured because of its exposure in the Mechanical application.

In addition, the following enhancements have been made to the curing module of ACCS:

- The surface exporter now supports additional formats such as `.inp` and `.cdb`. This enhancement is useful when you need to use a deformed shape with process-induced distortion for further postprocessing.
- Bug fix to support systems with non-dot decimal separators.

1.9. Envyo: Multi-purpose Mapping Tool

A new release of Envyo, the multi-purpose mapping tool closing the simulation-process chain for LS-DYNA, is available. This release comes with the following new features and enhancements:

THESEUS-SOLID

- Model-free kinetics evaluation has been added to calculate the degree of cure of adhesive materials after temperature treatment. You can use the results directly for a subsequent analysis with *MAT_307, or for initializing locally varying initial yield stress, hardening, and damage strain thresholds in *MAT_252 (IHIS=6).
- Clustering methods based on historical variables have been added.

SOLID-SOLID

- The degree of cure can be mapped directly from a previous curing analysis using *MAT_307, and then transformed into locally varying initial yield stress, hardening, and damage strain thresholds in *MAT_252 (IHIS=6). See also **THESEUS-SOLID**.
- Clustering methods based on historical variables have been added.

MAPPING FROM MOLDING SIMULATION

- Support for history variable initialization for *MAT_303 has been added for all molding simulation result data mapping routines.

1.10. PyACP Officially Released

You can now use ACP preprocessing features as a public Python module, , in a standard Python environment outside of Workbench. PyACP can be used in combination with other PyAnsys modules, such as PyMAPDL and PyDPF Composites. This enables the implementation of automated simulation workflows such as optimization and parameter studies. Since this is the first PyACP release, a few features or interfaces are not implemented 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 .

1.11. Updated Short Fiber Composites Tutorial

The second Short Fiber Composites tutorial has been updated to include the following:

- An explanation of required data for calibration.
- The recommended checks throughout the calibration process.
- How to plot the fiber-orientation tensor in global coordinates.

For more information, see Calibration and Validation of the Elasto-Plastic Properties of a Short Fiber Composite Tutorial in the *Short Fiber Composites Tutorials*.

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 sections present the notes for the Ansys Material Designer 2025 R1 release.

1. Notes

Parasolid Kernel as Modeling Engine

The Material Designer product now uses the Parasolid kernel as its geometry modeling engine, rather than ACIS. Therefore, generated geometries, meshes, and solutions may have slightly differing results between 2025 R1 and earlier releases.

Material Designer Installation

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

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

2. Known Limitations

General Material Designer product and release-specific limitations can be found in Known Limitations in the *Material Designer User's Guide*.

Additive Manufacturing

Additive Suite products include new features and enhancements at Release 2025 R1 in the following areas:

1. LPBF Simulations

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

- **Import AM models from a `.3MF` file**: You can now import an AM model consisting of part(s) and support geometries directly from a *3D Manufacturing Format*, or `.3MF` file. A new Import from 3MF button is available on the LPBF Process Add-on ribbon. During an import, Geometry Import objects are created and added in the Mechanical application's project tree, and an `.stl` file is generated for each part and support body and placed beneath the appropriate Geometry Import object. Additionally, an AM Process object is added, and the newly inserted geometries are automatically scoped to it. That is, the part and support geometries are automatically identified and the supports for each part are scoped within a single Support Group under AM Process.

  The import function works by reading the structure of the `.3MF` file. A `.3MF` file describes geometry by listing the coordinates of vertices and constituent triangles, much like an `.stl` file. The `.3MF` file also defines *relationships* between geometries, such as between parts and their related supports. The **Import from 3MF** feature supports the *core* specification as defined by the 3MF Consortium, which includes information about the geometries and their relationships but not material or slicing information.

- **Improved convergence for base unbolting step**: Nonlinear stabilization is now on by default for base unbolting, base removal, and support removal AM process steps to improve convergence. See Stabilization in the *Mechanical User's Guide*, and Using Nonlinear Stabilization and **STABILIZE** in the Mechanical APDL documentation for more information.

- **Improved convergence for removal steps**: A new Weak Springs boundary condition object, similar to other structural support objects, is now available to loosely hold parts in place during removal steps. It can be added using the new Weak Springs button on the LPBF Process ribbon, or by insertion under the Static Structural system in the project tree.

Insert a Weak Springs object into an LPBF Static Structural simulation to prevent rigid body motion during post processing steps when the default stabilization is insufficient. In most cases, simulations should be able to reach convergence without this object, but it may be necessary on some occasions. When added, the object automatically scopes all nodes at the bottom of the part and support geometries. It attaches weak springs on the scoping (or a subset of the scoping) to hold the part in place without significantly affecting results.

- **Location-specific Strain Scaling Factors (Beta feature)**: With this beta feature, you can specify strain scaling factor (SSF) values that vary in space in AM Inherent Strain simulations. This allows you to account for variability in strains for different locations on the build plate. Previously, a single SSF was applied to all elements of the build. With the beta feature, the existing Tables/Table tree items can be used to specify varying strain scaling factors for X and Y locations. The Tables/Table functionality also permits importing a `.csv` file of strain scaling factors.

- **Directional recoater interference (Beta feature)**: The directional recoater interference capability is an extension of the current recoater interference result. With the new capability, the recoater interference result will be modified to represent a difference in recoater interference likelihood with different recoater directions. Only results on the surface of a part are adjusted based on the recoater direction.

2. DED Process Add-on

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

- **Multi-axis printing**: Simulation of a multi-axis process is now fully supported in the application. In the DED Process Wizard or in the G-Code Clustering object, under Print Direction, choose Multi-axis to enable the functionality. (In previous releases, the multi-axis option was called 3D (Beta).)

  The G-Code clustering technique for multi-axis simulation is completely new, resulting in substantially faster cluster generation (at least ~100x faster!), improved connectivity checking for stable solver performance, and more robust handling of complex geometries as compared to the Beta feature. The new connectivity check finds and corrects badly connected elements if they exist. The check is turned on by default. You can turn it off when testing simulation setup to increase cluster generation speed.

  Multi-axis simulations require a Cartesian mesh but you can set the Projection Factor to something other than 0 to improve the quality of contact for better convergence.

  It is useful to choose multi-axis simulation even with a single print direction when you have multiple parts with various layer heights to avoid any misalignment problems.

- **DED Process Wizard**: The DED Process Wizard includes built-in automated functions to make DED simulations as easy as possible, while the project tree workflow empowers those users who prefer more direct control. With each new release, we add more smart checking functions into the wizard, so we are strengthening our guidance to use the wizard whenever possible to set up your DED simulations. At this release, the wizard now detects whether there is shared topology among bodies in the model and adjusts settings accordingly. Shared topology between the part and the base body is recommended when you have a curved base. The wizard is particularly helpful for multi-axis simulations where several new Named Selections are required to generate clusters and facilitate contact creation—the wizard creates these Named Selections and the contact connection (DED_Contact) automatically.

- **Direct cluster manipulation using cluster output file**: The build settings of deposition rate, preheat temperature, and dwell time can already be controlled down to the individual cluster level by using the cluster settings table. As an alternative, you can now edit the output file, `DEDClusterOutput.json`, from the cluster generation process directly. Change the build settings or add, remove, or move elements among clusters.

  Use the new right-click options on the G-Code Clustering object to do this. Right-click the G-Code Clustering object and choose **Open DED Clustering Directory**. From there, edit the `DED-ClusterOutput.json` file as you wish. This is the output file from the cluster generation process. After editing, right-click the G-Code Clustering object again and choose **Update Clusters and Settings**. This action regenerates the Named Selections and updates cluster settings but will not rerun the clustering algorithm.

- **Controlling heating and cooling per cluster during the build process**: You can now control the additional build settings of power (for power based heating method), or process temperature (for temperature based heating method), and radiation emissivity and gas convection coefficients during the build for both the part(s) and the base using command snippets (Commands object) with specific DED parameters. This may be useful to lower power in areas where there may be overheating, say, or to adjust build conditions for different materials.

3. Sintering Process Add-on

Enhancements for sintering simulations include the following:

- **Sinter Calibration Wizard**: Calibrate the viscosity and sintering stress prefactors that drive your sintering material using the Sinter Calibration Wizard. (This was a Beta feature in previous releases.) An easy workflow guides you through identifying measurements, setting basic values, and iteratively running analyses to identify calibrated material properties.

- A Beta workflow is available for multi-stage calibrations. These are temperature dependent calibrations with multiple sets of sintering measurements.

- **Sinter Schedule**: You can now export a sinter schedule as an `.xml` file and import a previously exported sinter schedule. These are right-click options on the Sinter Schedule object.

4. Distortion Compensation Add-on

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

- **Export compensated CAD geometry (Beta)**: This Beta feature allows you to transform/morph the original CAD geometry using the compensated `.stl` file. Internally, this feature uses a Discovery API (the Ansys Discovery™ application must be installed) to perform geometry transformation. At least one successful iteration on Iterative Compensation or a Single Compensation result object is required to generate modified CAD geometry.

5. Changes in Product Behavior

Release 2025 R1 of the Mechanical application includes AM-related enhancements that cause product behavior to differ from earlier releases, as explained below:

Laser Powder Bed Fusion (LPBF)

- Nonlinear stabilization is now on by default for base unbolting, base removal, and support removal AM process steps to improve convergence. This change may affect the deformation results in unbolting or removal cases, but the overall shape of the geometry is the same. See Using Nonlinear Stabilization and the **STABILIZE** command in the Mechanical APDL documentation for more information.
- Build to base contact for layered tet meshes was improved by increasing the tolerances on the Build Contact Element Faces named selection.
- For AM LPBF Thermal-Structural simulations using quadratic (higher-order) elements and the melting temperature option for heating (Heating Method on Build Settings), the code was changed to increase the heating time on the first layer. This should generally have negligible effect on the overall results but improves potential instabilities on the first heating step that sometimes led to artificially high temperatures.

Directed Energy Deposition (DED)

- In the DED Process Wizard, the order of the mesh and cluster setup steps has been switched. Now, if Print Direction is set to Multi-axis in step 2 (cluster setup), the mesh type field in step 3 (meshing) becomes read-only and set to Cartesian.

Distortion Compensation

- The refaceting code was improved and should better capture fine features and sharp angles.

6. Advisory: Ansys Additive Application End of Life

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

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

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

- The Mechanical application, for Additive Print functionality. Several types of geometry-based AM simulations are available, including laser powder bed fusion (LPBF), directed energy deposition (DED), sintering, and distortion compensation.
- PyAdditive, for Additive Science functionality. PyAdditive allows you to experiment with process parameters for determining the best combination to use in laser powder bed fusion processes of metal parts. See the next section on PyAdditive (p. 83) in these Release Notes.

If you have any questions or need assistance, contact Ansys support.

7. PyAdditive

You can now use the type of features found in Additive Science in a public Python module called PyAdditive, a standard Python environment outside of the Ansys Additive desktop application. PyAdditive is a Python client library for the Ansys Additive server. The Ansys Additive server is distributed with the Additive option of the Structures package in the Ansys unified installation.

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

Sherlock

Introduction

This document provides a summary of all new features, product enhancements, limitations, and bug fixes in the 2025 R1 release of the Ansys Sherlock™ electronics reliability prediction software.

1. Known Issues

When launching on Windows, the Sherlock application may have issues retrieving system information. When this happened on earlier versions, the application exited before displaying the user interface, and the log file generated a single line similar to the following:

```
11:32:31.064 INFO: Sherlock Client Started
```

This issue has been reported for Windows 10, both for the Sherlock application and the 3D Viewer. The known root cause is the malfunctioning of the Windows Management Instrumentation (WMI). The Sherlock application has been updated so it now shows an error dialog when this issue arises. When you close the dialog, the application will finish loading. However, Ansys recommends you repair the WMI if you encounter the issue. Otherwise, you may encounter unexpected errors. Upgrading from Windows 10 to Windows 11 has resolved the matter for some users.

2. New Features and General Enhancements

2.1. New Features

- A new beta feature enables Thermal Mech Life Predictions for BGA components. It also enables the use and export of detailed BGA package models for the Thermal Mech analysis See the Sherlock Beta Features.
- A new temperature dependent calculator has been added to the Materials Editor and can be enabled under Beta features. See the Sherlock Beta Features.

2.2. General Enhancements

- Strain map support has been added to the Harmonic Vibration analysis, including both CSV and image files.

- The following new solder types have been added to the Solder Library:
  - 901SC
  - CVP-390V SAC305 (This is a licensed version of SAC305.)
  - SOP 91121 Innolot-89M3
  - CVP-390V Innolot
- Notifications between the Sherlock and Workbench applications have improved. The Workbench application now receives notifications via the gRPC bi-directional stream when the following updates or actions occur in the Sherlock application:
  - When the **Material Property** of a component, mount pad, or mechanical part is changed.
  - When the **PCB Model** type is changed (**PCB Modeling** tab in the FEA Analysis Properties dialog).
  - When the **Filtered Part Results** property is changed (**PCB Modeling** tab in the FEA Analysis Properties dialog).
  - When cutouts, mechanical parts, and/or potting regions are added or deleted.
  - When mount pads and/or mount holes are added, deleted, or moved.
- When processing the Random Vibration RST file, the Sherlock application now reads the Segalman stresses directly which improves performance.
- The Thermal Mech analysis now processes the results model directly from the RST file.
- The beta flag **Polygon Merge Performance** has been added to enhance the performance of polygon merging during Trace Model generation.
- For BGA components, ball model meshing has been improved. When you select ADVANCED for the **Ball Modeling** property, the ball mesh is more spherical, improving the accuracy of FEA Analyses.
- The RST file import feature has been optimized. Some processes now employ parallel processing. Also, you can use the **Number of CPUs** setting to specify how many threads are to be used: **Sherlock Settings > FEA > Engine Properties > Number of CPUs**. See Analysis Properties in the *Sherlock User's Guide*.
- FEA model generation performance is now improved when **Lead Modeling** is enabled and the CCA contains BGAs with **Ball Modeling** enabled. You will see the improvement when running an analysis or exporting a model.
- When processing ODB++ files, the Sherlock application now attempts to determine the compression algorithm used for the ODB++ layers when the algorithm is different than what is stated in the specification format.
- Response time has been improved when stopping an analysis task.
- The Sherlock application has a new FEA setting called **Copy Imported RST File**. When the setting is DISABLED, the application accesses the imported RST file from its *source* location instead of copying it into the module's working directory.

- To reduce memory usage, the Sherlock application now clears in-memory cached results for the Mechanical Shock and the Random Vibration analyses when clearing other cached analysis results.
- When exporting a Parts List, there is now a check box that enables you to exclude non-required property values. The check box is only accessible when **Unused Part Properties** is set to HIDDEN in Advanced Settings. The check box is enabled by default. If you choose to export all columns, the columns pertaining to non-required properties will be blank in the output file.
- You can now prevent non-required part property data from being uploaded to the Local Parts Library by setting **Unused Part Properties** in **Advanced Settings** to **Hidden**.
- When viewing component layers in the Layer Viewer and the **Leads** filter is selected, the ball number for all BGA's are displayed when you zoom into the part.
- You can now control the sizing of the left-side filter panel of the Layer Viewer. To ENABLE this feature, click the **Settings** menu and then **Advanced**. Under **Layer**, enable **Layer Viewer Divider**.
- There is a new beta feature called **Solder Fatigue BGA Model Updates**. See the Sherlock Beta Features. When enabled, it updates the current method of computing BGA Solder Fatigue:
  - It splits the energy into hot and cold energies for intermediate calculations.
  - PCB Foundation Stiffness uses PCB Resin Properties.
  - It uses a stiffness knockdown for solder stress that is based on the PCB Foundation Stiffness and the Solder Shear Stiffness ratio.
- A console warning now appears on startup when Sherlock has been configured to use memory that exceeds available RAM.
- The performance of processing copper layers during ECAD import has been improved.
- When duplicating an Event in the Event Editor, the Sherlock application now creates a default name (event name + incremented number) for the new Event instead of leaving the **Event Name** field blank. The default name remains editable.
- When exporting a Trace Reinforcement Model on Linux, you no longer have to create a cross-section group manually beforehand. The exported model now completes this step automatically.
- Updated generated FEA models for completely circular parts which lie on the PCB surface: The parts are now rotated so that a complete part face is parallel to the PCB surface, allowing proper contacts to be generated.
- A new FEA setting **Clear Named Selections** has been added which, when enabled, will always delete the current Named Selection list in in the Mechanical application (including items added in the Mechanical application) before reinitializing it with the exported Sherlock data.
- The tooltip help text for Outline Simplification has been corrected.
- The Strain Mapping help menu typo has been corrected.
- The use of Strain Maps has been added to the Harmonic Vibe analysis.

- Error messaging has been improved when you specify a frequency range that does not contain any natural frequencies. The message now provides constructive feedback.

3. API Updates

Links:

3.1. New APIs

- **exportProject() API**: Exports selected project data/files into a `.zip` file.
- **updatePartListValidationProps()**: Sets properties for the Part List Validation analysis.
- **addModelingRegion()**: Adds a list of modeling regions for display in a specified CCA's Layer Viewer.
- **createCCAFromModelingRegion()**: Creates a list of CCAs from already existing modeling regions
- **updateModelingRegion()**: Updates a list of existing modeling regions in a specified CCA's Layer Viewer.
- **updatePottingRegion()**: Updates the fields in pre-existing potting region(s).
- **deletePottingRegion()**: Deletes pre-existing potting region(s).
- **copyPottingRegion()**: Copies pre-existing potting region(s).
- **deleteModelingRegion()**: Deletes a list of existing modeling regions.
- **copyModelingRegion()**: Copies a list of existing modeling regions.
- **getPartsListValidationProps()**: Obtains the Parts List Validation analysis properties for a CCA.

3.2. Modified APIs

- Error messages `Missing cca name` and `Missing cca description` have been changed to `Missing CCA name` and `Missing CCA description`.
- When a Part Library does not exist when running the **updatePartsList()** API, the error message has been changed from

  `Part library does not exist`

  to

  `Part library does not exist: <part library name>`

  where `<part library name>` is the name of the Part Library used when invoking the API.

- Moved **MatchingMode** enum {} definition from Parts Service to Common Service.
- Added the ability to get the Sherlock version and default directory to a Sherlock API.
- Updated **updateNaturalFrequencyProps()** API to reject **naturalFreqMin** and **naturalFreqMax** if integer values are not contained in both of these fields.

3.3. Deprecated APIs

- The nested PottingRegion message `AddPottingRegionRequest.PottingRegion` is being deprecated. It will be moved outside of the **AddPottingRegionRequest** message. It will then be referred to as **PottingRegion**, not **AddPottingRegionRequest.PottingRegion**.

4. Bug Fixes

- When importing an EDB file or exporting to Electronics Desktop, the Sherlock application sometimes used the wrong licensing client and so was unable to perform these functions. This issue has been corrected.
- Fixed a bug which prevented the end-user license agreement from being displayed in the **About** dialog.
- The **Parts List Validation Properties** dialog now sets the **Part Library** selection to the Part Library that was saved for the CCA.
- Corrected an issue with **addPottingRegion()** API where the rotation was not being used during the drawing process.
- For Random Vibration imported analyses, damping values entered on the analysis input form were not used unless previously saved and used. That has been corrected.
- When ball modeling is set to ADVANCED, the Sherlock application now sets all contact bonds between the solder ball mesh and the part. Previously, contact bonds were set only for four points of one of the mesh faces where the top of the ball contacts the bottom of the part.
- When using an API to run a Thermal Mech Analysis, Sherlock was not clearing previously selected Events and, so, was using all previously selected Events when running the **runAnalysis()** API. Now, Sherlock runs only those Events specified in the **runAnalysis()** request.
- When editing an existing mount hole for which no material had been set, an error dialog appeared when attempting to save the mount point. This issue has been corrected.
- Updated the material names assigned during Workbench export when using part weight to compute density for a material. When multiple parts with the same material but different volumes or weights were exported, only the last part with a given material was created.
- Fixed a bug that occurred when exporting a trace reinforcement model with the **Use FEA Model ID** option selected. It caused a script error in the Mechanical application when materials were assigned to the model.
- Fixed an issue with Advanced Lead Meshing in which the nodes of certain bent leads could be specified in an incorrect order, resulting in "inside-out" elements.

- Previously, when exporting a model with a modeling region that is completely contained within the PCB where the coordinates of the modeling region had a different resolution than coordinates of the cut area, this resulted in a small gap between the modeling region and the PCB. This has been corrected.
- Corrected a bug that sometimes happened when exporting a STEP model with a modeling region that overlapped the PCB outline and shared some interior edges with the PCB. The section of the PCB the modeling region overlapped would be missing from the export.
- When inputting values for **Min** and **Max Frequency** for the Natural Frequency Analysis, you are now only permitted to enter and save integers. This resolves the issue where decimal values resulted in the "Invalid Numeric Value" message in the displayed results.
- When you rename a project, the Parts List file location is now updated as well.
- When a round part was lying on the PCB surface without a part face parallel to the surface, the part would not be properly attached to the board. This has been corrected.
- When Sherlock attempts to find pads under a part with leads outside the PCB outline, this no longer results in an exception when exporting the CCA to AEDT.
- Corrected an issue which prevented CC packages with custom names from using the intended Weibull beta. When determining the Weibull beta used by a given part solder, the assigned package type is no longer based on the Package Name but on the assigned Package Type.
- Previously, Sherlock would not launch on Windows when there were issues retrieving system information. Now when this issue occurs, the system error is outputted to the console and an error dialog appears.
- When launching, Sherlock no longer hangs when a leadBend value is not set in the `<project name>/<CCA name>/_METADATA/partslist.csv` file.
- Updated the splash screens of the Sherlock auxiliary applications so their size is now consistent with that of the main Sherlock application.
- Updated the generated Trace Model file included in the Tutorial Project for `copper-06.odb` in the Main Board. The previous file had incorrect data which was evident when viewing the Trace Model.
- When drawing polygons for potting and modeling regions in the Layer Viewer, clicking-and-dragging no longer causes an exception.
- When creating a polygonal potting or modeling region in the Layer Viewer, doing a shift+double-click no longer creates two points. This was happening because double-clicking does a pan-and-zoom. Now, the Sherlock application adds just the first point instead of adding a point prior to and after the pan-and-zoom.
- Corrected an issue that caused the following failure in the DFMEA Export:

```
java.lang.NoClassDefFoundError: org/apache/commons/codec/digest/Diges-
tUtils
```

- When adding a polygonal potting or modeling region in the Layer Viewer, performing a Ctrl+click-and-drag before adding any points for the polygon no longer causes an exception.

- On rare occasions, when processing multiple copper layers simultaneously, the Sherlock application would not assign copper percentages. This has been corrected.
- When editing the mount points or potting region for the CCA of a Detailed Component, the Sherlock application would incorrectly show units as **in**. This has been fixed so the proper units are now displayed.
- The Sherlock application no longer hangs when attempting to view an image file with the following extensions `.tiff`, `.bmp`, `.gif`, or `.jpeg`.
- Corrected an issue where the Python script that initializes Named Selections in the Mechanical application allowed items with the same name to be added to the list.
- Corrected a bug that happened when opening a project in single-project mode: When double-clicking the project in which copper layers were not processed, this caused the copper layer contents to be displayed instead of the Layer Viewer representation of the copper layer.
- When processing a copper or laminate image layer, the Sherlock application would compute the copper percentage of the layer to be 100% if the background color of the image was not transparent. This has been corrected.
- When running the Update Laminate API, you can now enter a non-default value for the glass construction resin percentage. Previously, non-default values caused the API to fail.
- During PCB mesh creation with layered elements and copper image layers, the application assigned the same material to the PCB instead of separate materials per element when the image used a non-transparent background. This issue has been corrected.
- The Parts List **Failure Class** field no longer displays **N/A** in the Part Editor when the source is **Guess**.
- The functionality of the Parts Editor has been improved so that tab and field enablement/visibility are set properly when control values change. This includes when you change the **Part Type** or change one of the **Source** values. Previously, changing the **Part Type** changed the enabled/visible state of the tabs but not the controls in the tabs. And changing one of the **Source** values only changed the value of the corresponding control without changing the enabled/visible state of the tabs or other controls.
- When editing multiple stackup conductor layers, the application no longer assigns conductor thickness units to disabled fields.
- In the **Edit Assembly Properties** dialog, the center X and Y values are now converted when you change the units in the dropdown menu.
- Corrected an issue that occurred when editing assemblies where the application would render an assembly incorrectly in the Layer Viewer after a change in units.

Forming

The 2025 R1 release of the Forming application includes the following new features and enhancements:

- A new advanced solution type called Springback Compensation enables you to run multiple iterations of the simulation and automatically modify selected tooling to compensate for expected springback, and create final parts that are within defined tolerances of a target shape.
- Drawbead features have been enhanced. You may now define a drawbead profile and calculate the restraint and uplift forces automatically. You may now also generate a 3D drawbead mesh automatically, adding it to the project geometry, and fully simulate drawbeads effects rather than using a representation.
- With this release, the software supports a new project type, "Hot Forming". This type of project utilizes additional thermal settings and material properties to simulate Draw and Trim operations in a hot forming process. Temperature components of the simulation are viewable in Contour plots during postprocessing Analysis.
- A new "Legend Background" option displays a semi-transparent background behind the countour legend in the Graphics Window for improved visibility when the legend and project graphics overlap.
- Forming now supports display themes in the Graphics Window of the software. Choose between "Classic", "Light", and "Dark" themes in the **System Configuration** panel under **Graphics** → **Theme** menu.
- The Forming user interface now supports CATIA-style mouse controls. To enable this control mode, open the **System Configuration** panel, set **System** → **Keymap** → **control_mode** to **`Catia`**, and click **Apply**.
- Easily reverse the normal direction of a tool mesh using the new **Reverse Normal** command. Right-click a tool button (item C in Figure 4.2: Drawing operation task panel) in an operation's Task Panel to open the context menu with this command.
- Refine part meshes without leaving the application by using the new Refine Mesh feature that is available on the Mesh Check Task Panel.
- If part shrinkage after removal from the die is a concern in your process, a new **Tool Scaling Task Panel** available through the OP Setting Task Panelenables you to scale the size of a die by a percentage, and set a custom coordinate system for the scaling if desired.
- The Contour plot in the Analysis Tab includes new displayable components for strain differential and temperature.
- The Skidmarks tool in the Analysis Tab has been enhanced with new threshold parameters for more accurate display of possible surface scratches.

- The Section Cut feature has been enhanced for improved usability. An arrow on the section cut thumbnail view of the project shows the direction of view. Use the left and right arrow keys on your keyboard to move the section cut forward and back through the project geometry.
- The Identify tool now enables you to identify Parts, as well as the Nodes and Elements that were possible in previous releases.
- During Analysis, if you have two or more projects open and displayed simultaneously, choose **View Sync** from the **Options** tool to synchronize the orientation so that changes made in one pane also apply to the other panes of the graphics window.
- The application now displays the current unit system in the title bar, for easy reference.

Motion Release Notes

The following sections present the new features, enhancements and changes for the Ansys Motion™ standalone Solver, Preprocessor and Postprocessor for the 2025 R1 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

Release 2025 R1 includes several new features and enhancements that result in product behaviors that differ from previous releases. These behavior changes are presented below.

- The contact search algorithm for entities with infrequent contact has been enhanced by optimizing the multi-thread strategy. This improves solving performance for **Links** models.
- The moment equilibrium about the origin of the element reference frame has been enforced to prevent excessive force and moment at a ground-to-rigid body connection, even though all finite element (FE) related connections exist within the rigid body.

2. New Features

The following new features are available:

- **Tie Contact** results now output Normal Force, Penetration, Friction Force, Stiction Slip and Force Magnitude to the Motion Postprocessor.
- Coordinate system information from a Mechanical model is transferred to the standalone Post-processor.
- Pole-free sections and asymmetric options have been added to the pole design methods for **FE Film** generation.

3. Functional Enhancements

The following functional enhancements are available:

- The Ansys Motion **Premium** license now includes access to all general Motion capabilities except toolkits. Access to toolkits (Links, Drivetrain, Car, and EasyFlex) is now available with the Ansys Motion **Enterprise** license.
- **Animation** playback controls have been improved in the Motion Postprocessor user interface.
- The **Color Tablet** interface for background, contour and text in the Motion Postprocessor **Animation View Properties** has been enhanced.
- **Coordinate System Properties** in the Motion Postprocessor **Object Navigator** have been reorganized for clarity.
- Datasets for **2D Splines** in Motion Preprocessor **Interpolation Functions** no longer require at least 4 points of data.
- Instead of alphabetical sorting, natural sorting is adopted to display lists, following the current Mechanical convention.
- A dummy body at an RBE master point can affect the high-frequency eigenvalue solution when it is located far from the geometry. This effect has been minimized.

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 Release Notes

The following sections contain release information for Ansys Fluent 2025 R1.

Backwards Compatibility: Ansys Fluent 2025 R1 can generally read case files and data files from all past Fluent releases. Solver and model settings from previous case files are typically respected. However, in some cases due to defect fixes and core improvements to improve robustness and/or performance, convergence behavior and/or results obtained may be different. Such release-to-release changes are documented in the Fluent Migration Manual, along with instructions to recover the previous behavior when possible.

1. Supported Platforms for Ansys Fluent 2025 R1

Information about past, present, and future operating system and platform support is viewable via the Ansys website.

2. New Features in Ansys Fluent 2025 R1

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

2.1. Meshing Mode

New features available in the meshing mode of Ansys Fluent 2025 R1 are listed below.

Fluent Guided Meshing Workflows

- **General Improvements**
  - Performance of the Zoom in/out, Pan, and Rotate View graphics functionalities has been improved when used on very large models.
- The **Watertight Geometry** Workflow
  - In previous releases, when body labels are merged after generating a volume mesh or when cell zones are merged in the manage zones task, face zones originating from face zone labels

would have a "_1" suffix at the end of the name. In this release, that suffix has been removed. Note that this change will only apply for new cases created from scratch to avoid issues with backward compatibility of existing Fluent solver journals.

  - In the **Add Thin Volume Meshing Controls** task, you can now use the **Draw Thin Volume Regions** button to display thin volume mesh regions.

  - When thin volume meshing controls have been created, additional controls are available under **Global Thin Volume Mesh Settings**in the **Generate the Volume Mesh**. For details, see Generating the Volume Mesh.

  - When **selected-zones** or **selected-labels** is selected for the **Grow on** field in the **Add Boundary Layers** task, any label(s)/zone(s) that have already been assigned boundary layers will not be available for selection later in the **Add Boundary Layers** task.

  - In the **Add Boundary Layers** task, you can now use the **Create Spheres** option to create spheres for modification at invalid normals.

  - In the **Improve Volume Mesh** task, you can now specify quality criteria for multiple quality methods. For details, see Improving the Volume Mesh.

  - In the **Manage Zones** task, you can now separate face zones after the volume mesh has been generated.

  - Cylindrical and conical frustum refinement regions can now be defined in the **Create Local Refinement Regions** task.

  - `.scdocx` file types are now supported and can be imported in the **Import Geometry** task.

- The **Fault-tolerant Meshing** Workflow

  - Under the **Settings** drop-down button in the **Import CAD and Part Management** task, the **Refacet during loading** option is now available for geometries imported using the **DSCO** import route. When enabled, this option will refacet the geometry during conversion to FMD, thus eliminating the need for expensive post-load refaceting operations.

  - In the **Describe Geometry and Flow** task, the **Skin assemblies by wrap** option is now available for creating a wrapped surface mesh around the geometry (skinned surface mesh) either for external flow analysis or to be exported as part of a combined meshing workflow.

  - In previous releases, ACIS was used as the default import route for importing `.sat` and `.sab` files. In this release, `.sat` and `.sab` files are now imported using the Discovery (DSCO) import route.

  - The Parasolid import route has been added for importing `.x_t` and `.x_b` files.

- The **2D Meshing** Workflow

  - The **last-ratio** offset method type is now supported when adding 2D boundary layers. For details, see Adding 2D Boundary Layers.

  - In previous releases, ACIS was used as the default import route for importing `.sat` and `.sab` files. In this release, `.sat` and `.sab` files are now imported using the Discovery (DSCO) import route.

  - The Parasolid import route has been added for importing `.x_t` and `.x_b` files.

Mesh Generation

- When using the Rapid Octree mesher, the smooth mesh coarsening option is now supported as a full feature. This option causes the refined cells to better follow the geometry, and can reduce the overall cell count while maintaining a comparable level of accuracy. (Mesh Parameters)

2.2. Solution Mode

New features available in the solution mode of Ansys Fluent 2025 R1 are listed below. Where appropriate, references to the relevant section in the User's Guide are provided.

User Experience

- Interactive performance in the graphics window is improved for operations such as panning, zooming, and rotating, specifically for surfaces containing millions of triangles or faces.

- The performance of the graphical user interface is greatly improved for cases with thousands of zones, using models including Radiation, Multiphase, Species, Viscous, Energy, Battery, Eulerian. Boundary condition dialog boxes open significantly faster, initialization is quicker, and the overall setup time for cases is noticeably reduced.

- Drag and drop of graphics objects and plots across Fluent sessions is improved so that names of objects are preserved. Refer to Drag and Drop, Copy and Paste in the *Fluent User's Guide* detailed descriptions of the behavior.

Files

- You can export cases with periodic boundaries to CGNS and have a separate zone_t for each cell zone (`file/export/settings/cgns-separate-cellzones yes`), as long as periodic boundaries are slit prior to export (`mesh/modify-zones/slit-periodic`).

Solver-Meshing

- When defining a reference frame, you can now specify that the motion is defined by a motion definition. This means that the motion can be defined using expressions and/or input parameters. (Creating and Using Reference Frames)

- When defining the geometry for a deforming dynamic mesh zone with remeshing enabled, it is now possible to use an auxiliary geometry definition of the **Reconstruction** type.

- When deleting cells that are marked by a specified cell register, you can now choose to augment the list of cells, so that additional adjacent cells are added to the deletion list that would otherwise be isolated after deletion. This can help to ensure numerical validity. (Deleting Cells)

- The following improvements are now available when creating mesh interfaces using the default one-to-one interface method:

  - You can now select cell zones as part of the setup, which is the equivalent of selecting all of the wall and/or interface boundary zones associated with those cell zones. In some cases the selection of cell zones can simplify your setup, reducing the number of selections you

need to make and/or eliminating the need to identify all of the relevant boundary zones. (Using a Non-Conformal Mesh in Ansys Fluent)

  - When manually creating a mesh interface with a single pair of boundary zones, you can now enable the matching option, in order to avoid the creation of wall zones where the original zones did not match well. (Manually Creating One-to-One Mesh Interfaces)

  - You can now use a text command to specify that one or more pairs of boundary zones are to be excluded when one-to-one mesh interfaces are created. This can make your setup easier, as then you do not have to keep track of the unwanted pairs through each round of mesh interface creation. This text command applies whether you create mesh interfaces through the graphical user interface (GUI) or the text user interface (TUI). (Using a Non-Conformal Mesh in Ansys Fluent)

- In order to save setup and processing time, you can now use a single text command to create multiple copies of multiple cell zones, instead of making one copy of one cell zone at a time. When you copy a cluster of adjacent cell zones together, the face zones between the cell zones will retain their original boundary conditions, which can simply your setup. (Copying Cell Zones)

- Polyhedra are now a supported cell type for 2D meshes. When you read a 2D mesh that has hanging nodes / edges, the adjacent cells are now automatically converted to polyhedra by default. To revert to the behavior in previous releases, enter the following text command prior to reading the 2D mesh: `file/convert-hanging-nodes-during-read? no`. For further details about polyhedra, see Removing Hanging Nodes / Edges.

- When using mesh adaption, the polyhedral unstructured mesh adaption (PUMA) method is now available for 2D meshes, and is selected by default. This method does not create hanging nodes and requires less memory than hanging node adaption. To revert to the behavior in previous releases, enter the following text command prior to setting up the adaption: `mesh/adapt/set/method hanging-node`. (Refining and Coarsening)

Heat Transfer / Radiation

- The S2S radiation model is now available with the GPU solver.

- Support for GT-POWER has been updated to version 2024.2

- When using the pressure-based solver, you can now set the reference temperature used for the sensible enthalpy calculation. This can be useful when the specific heat is a strong nonlinear function of temperature, and the solution temperature range is far away from the default setting (298.15 K). Setting the enthalpy reference temperature to be within the solution temperature range may reduce roundoff errors when the enthalpy integral is computed and avoid unphysical oscillations of the temperature in the pressure-based solver. (Setting the Reference Temperature for Enthalpy)

Turbulence

- The Generic WMLES model, which uses LES for the bulk of the domain and the algebraic RANS model for the thin near-wall section of the boundary layer, is now available with the GPU solver. (Generic Algebraic WMLES: LES with a Near Wall RANS Layer in the *Fluent Theory Guide*)

- The Algebraic Transition and Curvature Correction models are now available for the k-omega SST and GEKO models with the GPU solver. (Algebraic Transition Model) and (Curvature Correction for the Spalart-Allmaras and Two-Equation Models)

Acoustics

- The **Ffowcs Williams & Hawkings** acoustics model is now available for the GPU solver. (Using the FW-H Model with the Fluent GPU Solver)

Turbomachinery

- You can now choose a **Reference Position** where the gauge pressure is applied when specifying a **Radial Equilibrium Pressure Distribution** on a pressure outlet. (Pressure Outlet Boundary Conditions)

- **Fast Mesh Smoothing** is now available when specifying periodic displacement for aerodynamic damping simulations. Fast Mesh Smoothing works by solving the mesh motion based on the periodic displacement at the beginning of the simulation, therefore not requiring the smoothing procedure after every timestep, saving significant simulation time. (Defining the Periodic Displacement of the Blades)

Discrete Phase Model

- For cases where the Lagrangian wall film model is used, a new postprocessing variable, **Wall Film Viscosity**, is now available in the **Wall Film...** category.

- For cases that involves the Lagrangian wall film model, the following discrete phase variables have been added:

  – `film-particle-htc-characteristic-length`

  – `film-particle-nusselt-number`

  – `particle-viscosity`

  See Particle Variables with the Lagrangian Wall Film in the *Fluent User's Guide* for more information.

- For cases involving the Lagrangian wall film model, you can now use the in situ reduction to decrease the number of DPM parcels representing the wall film. This was available as a beta feature in previous releases. (In Situ Data Reduction)

- For Selective Catalytic Reduction (SCR) systems, the level-based model for assessing the solids deposition risk has been added. The model computes risks associated with hydraulic, crystallization, and secondary chemical reactions. (Level-based Risk Assessment)

Multiphase Models

- For cases involving effects of the turbulent dispersion force, the following solver controls have been added to allow volume fraction limits to be adjusted for the limiting and transition functions via the text user interface (TUI):

  – Limiting function:

→ `/solve/set/multiphase-numerics/interphase-interactions/turbulent-dispersion/limiting-function/vof-lower-limit`

→ `/solve/set/multiphase-numerics/interphase-interactions/turbulent-dispersion/limiting-function/vof-upper-limit`

– Transition functions (Simonin and Burns et al. models):

→ `/solve/set/multiphase-numerics/interphase-interactions/turbulent-dispersion/transition-function/vof-lower-limit`

→ `/solve/set/multiphase-numerics/interphase-interactions/turbulent-dispersion/transition-function/vof-upper-limit`

See Including the Turbulent Dispersion Force in the *Fluent User's Guide* for details.

- For clarity, **kocamustafaogullari-ishii** that is available as the **Bubble Departure Diameter** model has been renamed as **kocamustafaogullari-ishii-bdd**, and **kocamustafaogullari-ishii** that is available as the **Nucleation Site Density** model has been renamed as **kocamustafaogullari-ishii-nsd**.

Electric Potential Field and Electrochemistry Model

- For the electrolysis and H2 pump model, the following enhancements have been implemented:
  – The ability to specify a reference temperature to calculate the reference current density in the Butler-Volmer equation. (Specifying Model Parameters (Parameters Tab))
  – The ability to plot capillary pressure against liquid volume fraction. (Specifying Porous Layer Properties for the Anode)
- You can now use anisotropic solid materials in your simulations. (Anisotropic Conductivity in Solids in the *Fluent Theory Guide*)
- The physics-based aging model is now available for the resolved lithium-ion battery model. (Lithium-ion Battery Aging Model)

Battery Model

- The method for detecting battery connections has changed from a zone-based to variable-based method. Defining separate active or passive zones for different batteries is no longer required. Active or passive zones associated with different batteries can now be grouped together, effectively reducing the total number of zones.

Proton Exchange Membrane Fuel Cell (PEMFC) Module

- You can now specify the Bruggeman coefficient for each porous media cell zone. (Setting Up the PEMFC Module)
- New user-accessible functions, `Dissolved_Water_Masstransfer (cell_t c, Thread *t, real *mt_gd, real *mt_ld, real *dS)` and `Phase_Change(cell_t c, Thread *t, real *mt, real *dS)` have been added. These functions allow you to customize mass transfer rates between liquid water, water vapor, and water. (User-Accessible Functions)

Solver Initialization

- You can now patch the initialized flow field using a named expression, rather than having to directly provide the expression in the **Patch** dialog box. Refer to Patching Values in Selected Cells in the *Fluent User's Guide* for additional information.

Fluent Native GPU Solver

- When defining solution monitors, only field variables that are supported by the GPU Solver will appear in the list of field variables.
- Solid time stepping is now available in the **Run Calculation** task page for transient CHT simulations.
- **Bounded Second Order Implicit** scheme for the **Transient Formulation** for incompressible time-dependant calculations. In addition, when enabled, the use**Use limiter in time** option becomes available for improving boundedness for transient thermal simulations.
- The following capabilities of Fluent solution mode are now supported by the Fluent GPU Solver:
  - Poor mesh robustness.
  - Report definition-based convergence conditions.
  - Flow rate for surface report definitions.
  - Expression report definitions.
  - **No Slip** condition for both stationary and moving walls.
  - Zero **Specified Shear** (free slip walls).
  - Turbomachinery flows:
    - → Full-wheel and multiple row turbomachinery models.
    - → Transient Rotor-Stator (TRS) with sliding mesh interfaces.
    - → Frozen-Rotor (FR) interface with multiple moving reference frames.
    - → Multiple row Periodic Sector simulations, including Periodic Instancing, for both FR and TRS models.
- CPU/GPU remapping is now supported as a full feature. Such remapping can be used to optimize the heterogenous parallel computing across CPUs and GPUs. It is beneficial to handle I/O operations of reading case and data and to conduct other CPU-intensive operations on CPUs, while it is performing the GPU-intensive computational tasks on GPUs. (CPU/GPU Remapping)

Density-based Solver

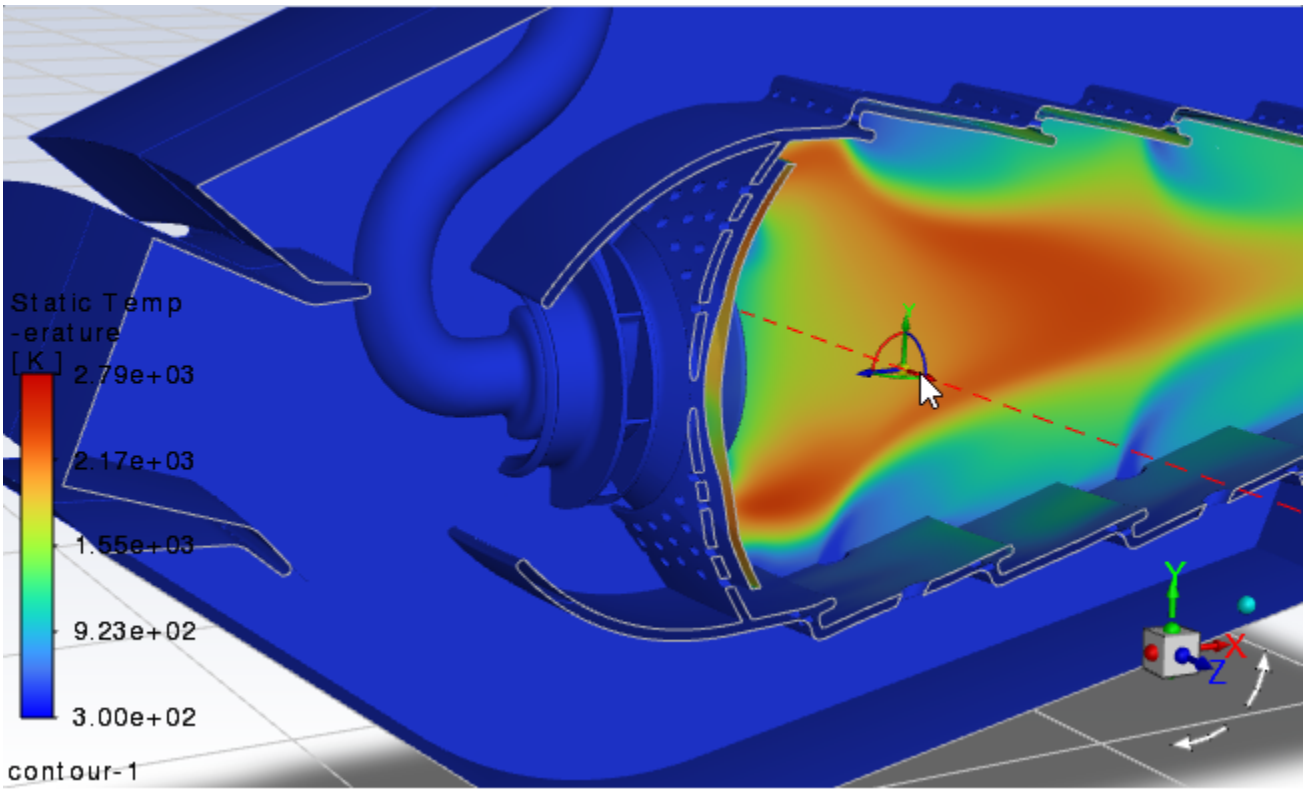
- Axis-stabilization for axisymmetric flows has been added to the density-based solver to increase the stability and accuracy of the solution near the axis boundary condition. It is automatically engaged when the high-speed-numerics is enabled. (Axis-stabilization for Axisymmetric Flows)

Graphics, Reporting, and Postprocessing

- Transient postprocessing provides the opportunity to further analyze the results of a transient solver run, allowing you to quickly generate animations, report plots, and result comparison plots from the data files produced. Refer to Transient Postprocessing in the *Fluent User's Guide* for additional information.

- A clipping plane is now available, allowing you to interactively see inside surfaces, as shown in Figure 1: Model Clipped with Cut Edges Highlighted (p. 106). This tool is accessed via the right-click context menu in the graphics window. Refer to Interactively Clipping the Display in the *Fluent User's Guide* for additional information.

Figure 1: Model Clipped with Cut Edges Highlighted

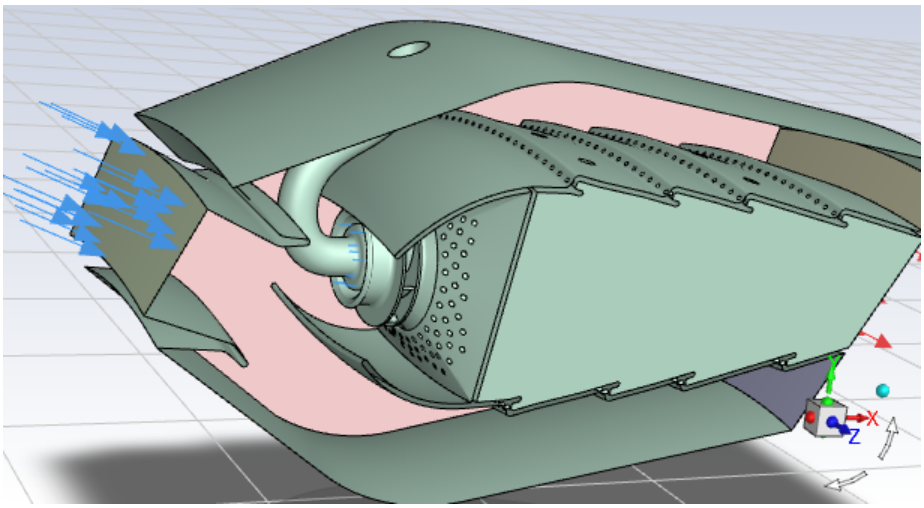


- Support is added for the OpenGL2 Mesa driver, which is equivalent in functionality to that of other shader drivers, but with reduced performance because the rendering is done without use of a graphics card (GPU). This allows for both basic and advanced graphics rendering on machines that do not have a dedicated graphics card.

- (Pastel colors) Mesh edges are darker than in previous releases, improving definition and visibility, as seen in Figure 2: Pastel Colors Mesh with Outline Edges Display (p. 107). You can display models with pastel colors by selecting **Pastel colors** from the **Model Color Scheme** drop

down in the **Appearance** branch of Preferences (**File**> **Preferences...**). *You can also specify an alternate color for edges in Preferences, if desired.*

Figure 2: Pastel Colors Mesh with Outline Edges Display



- You can now display multiple graphics objects at once in separate graphics windows either using the right-click context menu in the Outline View tree or by entering the `display/objects/display-in-different-windows` text command in the Console.
- The type of the graphics driver used for the solver run is now printed to the transcript.

Parallel Processing

- The `-stream` command line option is now available when running on Windows, so that you can print the memory bandwidth using a variant of the STREAM benchmark (instead of running Fluent). This utility is enhanced for all platforms to run a memory bandwidth scaling study, to help in determining if your memory is set up in an optimal manner. (Command Line Startup Options)
- When running on Linux, a new `infiniband.ofed` interconnect is available. (Starting Parallel Ansys Fluent Using Fluent Launcher and Starting Parallel Ansys Fluent on a Linux System Using Command Line Options)

Fluent for Arm

- When running Fluent for Arm, the integration of the CHEMKIN CFD Solver is now supported. For more details, see Running Ansys Fluent on Arm Compute Nodes.

Parametric Studies

- Parametric reports can now be exported as PowerPoint presentations. See Generating, Viewing, and Saving Your Parametric Reports for more information.
- The **Create Design Points and Optimize Using optiSLang** dialog now contains a **Use Start Designs Only** option that allows you to use existing design points as the starting points in your parametric optimizations. See Configuring optiSLang AMOP Settings for more information.

- For concurrent parametric runs, the **Concurrent Settings** dialog now allows you to submit remote jobs to an Ansys High Performance Computing (HPC) Platform Service cluster as well as configure local jobs. See Submitting Concurrent Parametric Jobs to Remote or Local Compute Resources for more information.

Cloud Computing

- You can now deploy your Fluent session to the Ansys Cloud directly from the Fluent interface so that you can run and/or monitor your CFD simulations remotely using the Ansys Cloud Burst Compute resource. See Performing Calculations Using Ansys Cloud Burst Compute for more information.

Note:

This feature is not available in Workbench, in the Fluent meshing workspace, or when using the Fluent solver workspace with GPU.

Expressions

- Scientific constants available for named expressions are updated in Fluent, reflecting the updated values as defined by the National Institutes of Science and Technology (NIST). The following constants are updated:
  - **R**—the gas constant. Updated to 8314.4626 $[\mathrm{kgm^2s^{-2}K^{-1}kmol^{-1}}]$.
  - **avogadro**—the Avogadro constant. Updated to 6.02214076e23 $[\mathrm{mol^{-1}}]$.
  - **echarge**—elementary charge. Updated to 1.602176634e-19 $[\mathrm{A\,s}]$.
  - **planck**—the Planck constant. Updated to 6.62607015e-34 $[\mathrm{J\,s}]$.
  - **stefan**—the Stefan-Boltzmann constant. Updated to 5.670374419e-08 $[\mathrm{W\,m^{-2}K^{-2}}]$.

Load Managers

- The graphical user interface (GUI) for the settings in the **Scheduler** tab of Fluent Launcher are now improved for usability. (Setting Parallel Scheduler Options in Fluent Launcher and Setting Job Scheduler Options When Running on Remote Linux Machines)
- When setting up the use of the Slurm job scheduler in the **Scheduler** tab of Fluent Launcher, it is now possible to specify a **Custom Option**. (Setting Parallel Scheduler Options in Fluent Launcher and Setting Job Scheduler Options When Running on Remote Linux Machines)

Web Interface

- As found in the standard Fluent interface, you can now interactively manipulate your line and plane surfaces directly in the graphics window. See Creating and Displaying Line Surfaces and Creating and Displaying Plane Surfaces for details.
- You can now use the web interface to create new and review existing periodic instances and display them in the graphics window. See Creating and Displaying Periodic Instances for details.

- You can run pre-existing saved (solution) animation files using the Results Arc and the **Play Solution Animations** tool where, when selected, any existing saved animation files will be listed and, when one of them is chosen, they will appear in a dedicated viewer. See Running Saved Animations for details.

- Calculation-based controls found in the Results Arc such as initializing and running a calculation now allow you to expand the panel to expose all available controls. See Initializing the Solution and Running the Calculations for details.

- For improved security when sharing web sessions, you are now able to specify permissions and more securely share your web sessions with others. See Setting Permissions and Sharing Your Web Sessions for details.

- When starting a web session using the Fluent Launcher or through the command line, you can now provide an optional email address where details about the web session and notifications can be sent. See Getting Started With Ansys Fluent's Web Interface for details.

- When displaying graphics objects (such as contours, vectors, and so on) during the course of running a simulation, you can control the refresh rate of such displays using the **Auto-refresh** toolbar. See Working With Graphics Objects and Your Simulation Results for details.

- You can now animate any displayed pathlines and particle tracks that may exist in your session. See Animating Pathlines and Particle Tracks for details.

- While creating certain surfaces (plane, and line surfaces), you can also interactively position them in the graphics window by clicking and dragging the arrow(s) and by manipulating the embedded triad(s) so that the surface is in the desired position.

- Several new models are available in the web interface, including Electrochemistry and various Battery models including SOFC and PEMFC. See Accessing Model Settings for additional information.

  In addition, there are several models in the Fluent desktop that are only available as add-on modules (such as battery-related models, for example) and require you to load the model using the text user interface. These such models are automatically made directly available in the Fluent web interface, with the exception of the **Fuel Cell and Electrolysis** model, which is only available in the Fluent desktop and is *not* available in the web interface.

- The property panels for several objects such as contours, vectors, pathlines are now presented with all available properties displayed by default using expandable headings. They will no longer use the **More...** and **Less...** exposure convention of showing and hiding additional fields.

- For any named objects (for example, setup-based objects such as boundary conditions, and results-based objects such as contours, vectors, mirror planes, and so on), a manager-based panel is now available so you can more easily manage such objects. For instance, in each case, you can view a list of similar objects and select one or more of them from a list. You can add new objects to or delete one or more existing objects from the list. In addition, you can edit the properties of one (or more) selected objects using the manager.

- Particle tracks are now available to assist in postprocessing discrete phase analyses. They are available in the Outline View as well as in the Results Arc.

- Contours now have **Coloring** options if you want to apply **Smooth** or **Banded** coloring to your contour plots.
- Additional features have been applied to color maps that include:
  - Spacing in and around the color map has been improved.
  - The positioning of labels has been improved when the color map is repositioned elsewhere in the graphics window (such as at the top, bottom, or on the left hand side).
- Additional features can be found in property panels that include:
  - For selection lists, just as in Fluent, you can view the number of selections versus the number of total available objects in the selection list.
  - For expandable headings, you can choose to expand and collapse all headings in order to show or hide the available properties displayed in the property panel.
- To improve the visual user experience throughout the interface, spacing, margins, and assorted padding improvements have been introduced for various panels and menus.
- You can now adjust the width and height, and even enlarge to full screen, certain portions of the interface, such as the console window and plot windows (such as XY plots, histograms, and cumulative plots). In addition, you can also now adjust the width of the outline view.
- Vector objects now include additional **Options** such as **Style**, and **Skip**, as well as **Vector Options** such as **In Plane**, **Fixed Length**, **Scale Head**, and the ability to set the *x*-, *y*-, and *z*-components of the vectors.
- Pathline objects now include an additional **Style** property (for giving pathlines different visual representation such as lines, points, spheres, etc.) as well as **Style Attributes** for a selected style. Ribbons, however, are currently not supported.

Beta Features

- There are also some exciting new enhancements available as beta features that you may be interested in trying out. Detailed documentation is in the *Fluent 2025 R1 Beta Features Manual*.

2.3. Fluent Workspace Applications

New features available in the client applications of Ansys Fluent 2025 R1 are listed below.

Fluent Icing

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

- **User-Defined Function (UDF) Access in the Client Ribbon**

  Particle drag and wall interaction models can be specified as user-defined functions. This allows the programming of custom models that may be proprietary or part of academic research. To facilitate UDF management, a **User-Defined** section has been added to the **Project** ribbon to

assist in building and loading UDF source files, setting user defined memory, and executing on-demand type functions.

- **View Ice Cover With Original Surface Display Using Viewmerical**

  The ice cover display feature has been improved to show the original clean surfaces (`map.grid 1`) as the secondary surface, rather than the `Fluent` surface of the current icing shot.

- **New Solver Numerics Defaults for the Pressure Based Solver (PBNS)**

  Fluent is a general-purpose flow solver and requires special tuning of solver parameters depending on the target application. To obtain accurate and robust results with Fluent Icing, especially for multishot with remeshing simulations, a set of solver defaults are now applied when solver settings are modified from **Case settings** to either **CFL** or **Pseudo-time** methods.

- **Multishot Icing With Anti-icing Heat Loads**

  The anti-icing heat flux distribution determined with an anti-icing CHT simulation can now be used as part of a multishot icing simulation with remeshing. The residual ice shapes can now be computed with multishot. This feature is especially useful when certain surfaces are partially protected, like the outboard sections of aircraft wings.

- **Ice Smoothing**

  The ice smoothing feature of FENSAP-ICE is now accessible in Fluent Icing under **Setup** → **Ice** → **Model** → **Ice Smoothing Iterations**. This feature conservatively smooths the ice shape to simplify it if needed. Originally intended for turbomachinery applications, where remeshing guide vanes bounded by hub and shroud walls can sometimes be challenging, it can also be used to reduce the mesh size of the iced grids in all icing cases while maintaining the bulk features of the ice shape.

- **DPM Ice Crystal and Water Vapor Two-Way Coupling**

  Vapor transport can now interact with the Lagrangian particle field solved by DPM in an iterative manner, updating ice crystal evaporative heat fluxes based on evolving local vapor pressure. This feature is accessible through the beta features flag.

- **DPM / ICE3D Coupling for Reinjection via UDFs**

  This feature allows you to program user-defined fields for particle-wall interactions, utilizing the surface icing solution as part of sticking, splashing, and bouncing modeling. Executed as a subset of the multishot algorithm, it enables the evolution of surface film height, ice thickness, and other factors to update the reinjection parameters of particle tracks. This feature is accessible through the beta features flag.

- **ICE3D Ice Crystal Icing (ICI) Model Exposure**

  The Ice Crystal Icing (ICI) model of FENSAP-ICE is now available in Fluent Icing. This 3-layer ice/water mixture model is designed to capture key physics of ice crystal icing in warm engine components, where the forming ice is considered spongy with a variable liquid fraction. The model can compute adhesion loss or constant accretion in the ice crystal icing regime on warm surfaces with a heat reservoir. This feature is accessible through the beta features flag.

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.

- **Aerodynamic Extraction Tool (AET)**

  – Proper usage of this tool now requires a separate installation of Ansys SpaceClaim, which is no longer a part of the full Ansys installation package. Consult AET: Aerodynamic Extraction Tool in the *Fluent Workspaces User's Guide* to properly install Ansys SpaceClaim.

  Averaging of Aerodynamic Coefficients (Beta)

  – Averaging of aerodynamics coefficients has been considerably improved. Averaging is now performed once steady oscillations are present within the convergence history of aerodynamic forces, therefore providing better estimates of average quantities as the transition phase is neglected.

  – This feature is available as a post-processing feature and gives the opportunity to the user to compute these values after simulation for all design points or a set of design points.

  – This feature has been incorporated into the Parameter Search workflow to improve convergence of the search algorithm when oscillating solutions appear within a cycle.

  – This feature has been incorporated into the AET workflow to provide more reliable averaged aerodynamic coefficients specially at high angles of attack.

- **Mesh Adaption (Beta)**

  – The usability of the **Adaption** feature was significantly improved. 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. This feature helps users to achieve accurate mesh independent solutions that capture detailed flow phenomena that may not have been possible using a baseline mesh over a wide range of conditions. The following points highlight some of the feature improvements in this release.

    → Cycle Parameter (CP) objects have been implemented which allow the Mesh Adaption tool to track progress of each adaption cycle.

→ Cycle *Restart* and *Continue to Update* functionality was implemented via the *Input: Design Points* table. In this manner, mesh adaption solutions can be restarted or continued from adapted or non-adapted cycles, thus offering much more solution flexibility.

→ Mesh adaption of 2.5D meshes are supported.

→ Parametric Post Processing of Cycle solutions is now possible for all mesh adaption simulations. This allows the user to generate images of meshes, contours, streamlines and scenes for each adaption cycle and all design points.

Aerodynamic Coefficients: Parameter Search (Beta)

– The Parameter Search feature, used to search for certain aerodynamic conditions, such as constant or maximum lift conditions, has been improved in this release.

– Cycle Parameter (CP) objects have been implemented which allow the Parameter Search tool to track progress of each search cycle.

– Cycle *Restart* and *Continue to Update* functionality was implemented via the *Input: Design Points* table. In this manner, parameter search solutions can be restarted or continued, thus offering much more solution flexibility.

– Parametric Post Processing of Cycle solutions has been extended to cover parameter search simulations. This allows the user to generate images of contours, streamlines and scenes for each search cycle and all design points.

Polyflow

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

- **Fluent Materials Processing Workspace Rebranded**

  The Fluent Materials Processing workspace has officially become the Polyflow workspace. This rebranding reflects our focus on expanding the capabilities of the well-known and reputed Ansys Polyflow Classic package in a new user environment.

- **Full Release of Tracking and Statistics Features**

  The **Tracking** and **Statistics** features are now fully released. The **Tracking** feature allows you to compute the pathlines of a set of material points. The **Statistics** feature allows you to perform a statistical analysis on a set of pathlines (evaluated in the **Tracking** task or coming from external mixing files).

- **Dissagglomeration Model as a Beta Feature**

  The Disagglomeration model has been introduced as a beta feature and allows you to model the disagglomeration of solid particles in a fluid, through erosion and rupture processes.

- **Update All Button Added**

An **Update All** button has been added to the graphical user interface to enhance usability. This feature streamlines the update process by applying changes across multiple parameters with a single click.

3. Updates Affecting Ansys Fluent 2025 R1 Code Behavior

The following sections list the code changes in Ansys Fluent 2025 R1.

General

3.1. Meshing Mode

This section contains a list of code changes implemented in the meshing mode of Ansys Fluent 2025 R1 that may cause behavior and/or output that is different from the previous release.

Meshing Workflows

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

Mesh Generation

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

3.2. Solution Mode

This section contains a list of code changes implemented in the solution mode of Ansys Fluent 2025 R1 that may cause behavior and/or results that are different from the previous release.

Files

- The significance of the names of the export file formats accessible via the `file/transient-export/particle-history-data` text command has changed:
  - The former `geometry` file type is now called `history`. The file name extension for files written by this option has been changed from `.curpart` to `.dpmhist`, and the iteration / time step placeholders `%i` and `%t` in the file name are now properly expanded.

  – The `geometry` file type now exports a file in the imported blend data file format (file name extension `.ibl`).

Solver-Meshing

- By default, the non-overlapping zones associated with one-to-one mesh interfaces are now shown only under their boundary condition types in the outline view tree (for example, under **Setup** / **Boundary Conditions** / **Wall**); if you want them to instead be shown only under **Setup** / **Mesh Interfaces** (under the name of their mesh interface), you can enable the **Show interface non-overlapping boundaries** option in the **Appearance** branch of the **Preferences** dialog box (accessed through **File**/**Preferences...**).

- When using a dynamic mesh with the `mesh/show-periodic-shadow-zones?` text command enabled (which is the default), the handling of periodic boundaries is improved so that all geometry options are now available for node projection. As a result of this improvement, when you define a dynamic mesh zone for a periodic boundary, you must now also define one for it's shadow. For further details about dynamic meshes, see Using Dynamic Meshes.

Solver-Numerics

- The default porous interface treatment is now improved when using the physical velocity formulation for compressible flows, in order to ensure the correct prediction of the pressure in the porous zone. This improvement is applied when interior face zones exist between two cell zones using different porosity and/or between cell zones in which only one zone is porous, and replaces the following Scheme command: `(rpsetvar 'porous/interface-incompressible-treatment? #t)`. If you do not want to take advantage of this improvement, you can undo the default change through the appropriate prompt in the following text command: `solve/set/previous-defaults/undo-2025r1-default-changes?`.

- The calculation of the Green-Gauss Node Based (GGNB) gradient is improved for User-Defined Scalars, Lithium, Potential, and/or Water Content models, in order to increase the accuracy. If you do not want to take advantage of this improved accuracy, you can undo the GGNB gradient enhancement through the appropriate prompt in the following text command: `solve/set/previous-defaults/undo-2025r1-default-changes?`.

Turbulence

- For the WALE LES model, the turbulence length scale is no longer limited close to walls, as such limiting was found to have a poor effect on the tendency of the flow to separate. As turbulent boundary layer flow separation is a main driver for LES models, the limiter is now disabled. If you do not want to take advantage of this improvement, you can undo the deactivation of the limiting through the appropriate prompt in the following text command: `solve/set/previous-defaults/undo-2025r1-default-changes?`.

- All turbulence specification methods at outlets are now supported with Fluent GPU 2025 R1, such as backflow conditions for a pressure outlet boundary. In Fluent GPU releases prior to 2025 R1, only the option **Intensity and Viscosity ratio** was available as a turbulence specification method at outlets. Reading an existing `.cas` files with an unsupported turbulence specification option automatically converted to **Intensity and Viscosity ratio**. This conversion is no longer performed in 2025 R1, and Fluent will use the method specified in the `.cas` file.

- For turbulence models based on the transport equation of the turbulent dissipation rate, the wall cell extrapolation of the turbulent dissipation rate from the cell center to the wall face

for the GPU solver was consolidated with the Fluent-CPU solver. The 2024 R2 Fluent-GPU solver behavior can be recovered by using the following command:

```
(rpsetvar 'gpuapp/use-epsilon-wall-extrpln-bc? #f)
```

Discrete Phase Model

- Minor changes have been made to the Lagrangian Wall Film model implementation that improve the solution stability and result in smoother appearance of the wall film temperature prediction. This may lead to slight differences in the solution results.
- The `P_...()` macros are no longer supported. Instead, use the `TP_...(tp)` macros for the `Tracked_Particle *` argument type and the `PP_...(pp)` macros for the `Particle *` argument type.
- For cases that involve the tabulated diameter distribution with the **Interpolate Between Classes** option, cubic spline interpolation is now used between user-defined diameter classes instead of the previous linear interpolation scheme. This results in smoother distributions that better approximate user-defined distributions.
- For DPM cases with the **Dynamic Interaction Range** option, the formulation of the particle specific length scale has been improved and made consistent for steady and transient simulations. This may lead to significant changes in the results, depending on the particle length scale and the mesh resolution.

Multiphase Models

- For the Burns et al. turbulent dispersion model, the default value of the model constant has been changed from 1.0 to 0.8333 to ensure agreement with the original Burns et al. model.
- The formulation of the Troshko-Hassan turbulence interaction model has been corrected in the documentation to match the implementation in the code. See details in Troshko-Hassan in the *Fluent Theory Guide*.

Electric Potential Field and Electrochemistry Model

- The solver robustness of the resolved lithium-ion battery model has been significantly improved. As a result, the number of iterations per time step in the transient simulation can be as low as 20.

Battery Model

- A variable-based battery connection detection method has replaced the zone-based method. Now, different battery active or passive zones can be grouped together, reducing the total number of zones in a simulation. This could lead to noticeable improvements in simulation performance.

Parallel Processing

- The ability to specify the use of General Purpose Graphics Processing Units (GPGPUs) in offload mode is now removed. This means that it is no longer possible to use such GPGPUs to accelerate the Algebraic Multigrid (AMG) solver, the calculation of view factors with the Surface-to-Surface (S2S) radiation model, and/or the calculations for the Discrete Ordinates (DO) radiation model. It is recommended that you instead accelerate such calculations by using the Fluent

native GPU Solver (as described in Using the Fluent Native GPU Solver). Note that the acceleration of the DO radiation model can still be done with CPUs, and is supported with the GPU Solver as a beta feature.

Graphics, Reporting, and Postprocessing

- The **Clip to Range** option in **Contours** dialog box is no longer automatically enabled when the **Auto Range** option is disabled.
- The ability to read keyframe scene animation video files in MPEG (`.mpg`) format is removed. Saving keyframe scene animations in `.anm` format will allow you to read them back into Fluent.
- The evaluation of a report definition for drag, lift, moment, or force by the Fluent native GPU Solver is now significantly faster when you are monitoring many zones and you have enabled the **Per Zone** option (to report the data from individual wall zones rather than the net results from a group of wall zones). For more details on such report definitions and the GPU Solver, see Force and Moment Report Definitions and Using the Fluent Native GPU Solver, respectively.

Expressions

- Scientific constants available for named expressions are updated in Fluent, reflecting the updated values as defined by the National Institutes of Science and Technology (NIST). These changes will affect expressions using these constants in Fluent2025 R1 when compared to previous releases. The following constants are updated:
  - **R**—the gas constant
    - → Previous value: 8314.4725 $[\mathrm{kg m^2 s^{-2} K^{-1} kmol^{-1}}]$
    - → 2025 R1 value: 8314.4626 $[\mathrm{kg m^2 s^{-2} K^{-1} kmol^{-1}}]$
  - **avogadro**—the Avogadro constant
    - → Previous value: 6.02214199e23 $[\mathrm{mol^{-1}}]$
    - → 2025 R1 value: 6.02214076e23 $[\mathrm{mol^{-1}}]$
  - **echarge**—elementary charge
    - → Previous value: 1.6021765e-19 $[\mathrm{A\,s}]$
    - → 2025 R1 value: 1.602176634e-19 $[\mathrm{A\,s}]$
  - **planck**—the Planck constant
    - → Previous value: 6.6260688e-34 $[\mathrm{J\,s}]$
    - → 2025 R1 value: 6.62607015e-34 $[\mathrm{J\,s}]$
  - **stefan**—the Stefan-Boltzmann constant
    - → Previous value: 5.670374e-08 $[\mathrm{W\,m^{-2} K^{-2}}]$

→ 2025 R1 value: 5.670374419e-08 $[\mathrm{W\,m^{-2}K^{-2}}]$

3.3. Fluent Workspace Applications

This section contains a list of code changes implemented in the client applications of Ansys Fluent 2025 R1 that may cause behavior and/or output that is different from the previous release.

Polyflow

- – The number of processors used by the workspace solver has been increased from 2 to 4 processors.

  – Mesh objects created by the workspace no longer use generic naming conventions (**mesh-1**, **mesh-2**, etc.) by default. The new default is to use the topological names found in the mesh file. The behavior can be changed using **Preferences**. See Setting Preferences for Polyflow in the *Fluent Workspaces User's Guide* for details.

4. Features To Be Removed in a Future Release

This section contains a list of features that are supported in Release 2025 R1, but will be removed in a future release of Ansys Fluent. It is recommended that you begin to migrate any cases that use these features at your earliest convenience.

CFX Release Notes

The following sections contain release information for Release 2025 R1 of Ansys CFX.

1. Supported Platforms

Platform/OS levels that are supported in the current release are posted on the Ansys website.

Note:

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

2. New Features and Enhancements

This section lists features and enhancements that are new in Release 2025 R1 of Ansys CFX.

- The **Table Generation** settings for the `IAPWS Library` **Material Properties** option have been extended to enable adaptive placement of table points and dynamic table ranges. Both of these features are aimed at reducing table size while maintaining resolution within an established error tolerance. For details, see Table Generation in the *CFX-Pre User's Guide*.
- There is support for modeling the effect of choking for the flow of a compressible fluid through the holes/slots of an injection region. For details, see Flow Regime in the *CFX-Pre 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 R2), and the support changes (functionality that is no longer supported) in Release 2025 R1 of Ansys CFX.

- For compressible fluids in rotating domains, the following rothalpy-frame variables are no longer included in the results file by default:
  - Total Pressure

– Total Temperature

– Total Density

This change was made in order to prevent numerical problems for high-angular-velocity rotating systems, where rothalpy may exceed the valid range for the material properties.

The variables can be included by selecting them from **Extra Output Variables List** on the **Results** tab of the **Solver** > **Output Control** details view of CFX-Pre.

Loading a project/state/session file from a previous release can lead to errors being generated if any variables that are no longer included in the output are referenced. It is possible to work around some of these issues by including the variables as mentioned above. In other cases, you might need to reset cells, regenerate reports, and potentially take other actions to eliminate references to variables that are no longer included in the output.

4. Updates Affecting Code Behavior

This section contains a list of changes that may cause the solution results from Ansys CFX to differ between Release 2025 R1 and Release 2024 R2.

- The RGP Generator used internally by CFX-Pre generates tabular data with a different distribution of table cells. Although the new distribution improves the resolution of gas property values, more table cells are generated for a given interpolation tolerance. To compensate for the larger number of cells, the `Low`, `Medium` and `High` **Table Resolution** options for the **Generate RGP File** tool (see Generate RGP File in the *CFX-Pre User's Guide*) have been adjusted towards lower resolution. Compared to the previous release, for a given interpolation tolerance, the resulting table files are of comparable size and have similar or better resolution of gas property values.

CFD-Post Release Notes

The following sections contain release information for Release 2025 R1 of Ansys CFD-Post.

1. Supported Platforms

Platform/OS levels that are supported in the current release are posted on the Ansys website.

Note:

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

2. New Features and Enhancements

This section lists features and enhancements that are new in Release 2025 R1 of Ansys CFD-Post.

Release 2025 R1 of CFD-Post has no new features or enhancements.

TurboSystem Release Notes

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

These release notes cover:

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

These release notes do not cover:

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

Note:

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

1. Supported Platforms

Platform/OS levels that are supported in the current release are posted on the Ansys website.

2. New Features and Enhancements

This section lists features and enhancements that are new in Release 2025 R1 of TurboSystem.

Release 2025 R1 of TurboSystem has no new features or enhancements.

TurboGrid Release Notes

The following sections contain release information for Release 2025 R1 of Ansys TurboGrid.

1. Supported Platforms

Platform/OS levels that are supported in the current release are posted on the Ansys website.

Note:

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

2. New Features and Enhancements

This section lists features and enhancements that are new in Release 2025 R1 of Ansys TurboGrid.

- You can select BladeBuilder, instead of BladeEditor, as the CAD generation tool to use when:
  - Importing NDF files (see Import NDF File Command in the *TurboGrid User's Guide*)
  - Creating CAD from profile points (see Geometry Browser Settings (CAD From Profile Points Mode) in the *TurboGrid User's Guide*).

  The default selection is controlled by preference **Default CAD Generation Tool** (see TurboGrid Options in the *TurboGrid User's Guide*).

  A BladeModeler or BladeBuilder license is required, depending on the selected CAD generation tool.
- Complex Blade End meshing parameters can now be specified by expression and can be linked to Workbench parameters.
- For curves displayed on the blade that correspond to the raw curve file data, and for the hub and shroud curves, arrowheads are added to show the direction of increasing point index number. For details, see Render Properties Show Curves Command in the *TurboGrid User's Guide*.
- A complex blade end is now allowed to extend upstream of the inlet end of the passage domain and/or downstream of the outlet end of the passage domain. For details, see the descriptions for the **High-fidelity Hub Features** settings in the **Complex Blade End** tab for a complex blade end `Geometry` object, in Complex Blade End Tab in the *TurboGrid User's Guide*.

A complex blade end has new `Mesh Data` settings that control the size of the surrounding tip shell. The applicability of these new settings is dependent on which **High-fidelity [Hub/Shroud] Features** settings are selected in the complex blade end's `Geometry` settings. For details, see the descriptions for the tip-shell-related settings in the **Complex Blade End** tab for a complex blade end `Mesh Data` object, in Setting Descriptions for Complex Blade End Mesh Data Objects in the *TurboGrid User's Guide*.

Ansys BladeModeler Release Notes

The following sections contain release information for Release 2025 R1 of Ansys BladeGen, Ansys BladeEditor, and Ansys BladeBuilder.

1. Supported Platforms

Platform/OS levels that are supported in the current release are posted on the Ansys website.

2. BladeGen

2.1. New Features and Enhancements

This section lists features and enhancements that are new in Release 2025 R1 of Ansys BladeGen.

Release 2025 R1 of BladeGen has no new features or enhancements.

3. BladeEditor

3.1. New Features and Enhancements

This section lists features and enhancements that are new in Release 2025 R1 of Ansys BladeEditor.

Release 2025 R1 of BladeEditor has no new features or enhancements.

4. BladeBuilder

4.1. New Features and Enhancements

This section lists features and enhancements that are new in Release 2025 R1 of Ansys BladeBuilder.

- There is a BladeBuilder component system in the Ansys Workbench toolbox.
- BladeBuilder can be started from the **Start** menu in Windows.

- BladeBuilder can make approximate blends for use in Ansys TurboGrid. For details, see BladeBuilder Settings in the *TurboSystem User's Guide*.
- There is a new Geometry Creation setting: Tip gap extension factor. For details, see BladeBuilder Settings in the *TurboSystem User's Guide*.

Polyflow Classic Release Notes

The following sections contain release information for Ansys Polyflow Classic 2025 R1.

1. New Features

The new features in Ansys Polyflow Classic 2025 R1 are as follows:

- **Transition to Polyflow Classic**

  Polyflow has been officially rebranded as Polyflow Classic. This change highlights our dedication to providing a robust and familiar platform for our users.

- **Non-Isothermal Model of Calibrator**

  A non-isothermal model for the calibrator has been introduced, allowing to model heat exchange between the extrudate and the calibrator, and in the case of contact between the fluid, and the solid and when the fluid is in close vicinity of the calibrator.

- **Improved Matrix Building Performance**

  The performance of matrix building has been significantly enhanced, especially for large-size calculations.

- **Enhanced Support for Windows Clusters**

  Support for Windows clusters has been improved, enabling more effective use of distributed computing environments.

- **Retirement of Polyman**

  The Polyflow Project manager utility, Polyman, has been retired.

- **Removal of CFF Output Beta Feature**

  The CFF output beta feature has been removed from this release.

2. Supported Platforms

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

3. Resolved Issues and Limitations in Ansys Polyflow Classic 2025 R1

This section lists limitations that existed in previous releases, but that were removed in Ansys Polyflow Classic 2025 R1.

Forte Release Notes

The enhancements and defect corrections listed below are for Release 2025 R1.

1. New Features and Enhancements

This section lists new features and enhancements in Release 2025 R1 of Ansys Forte CFD, organized by topic.

Documentation

- Updated Composition Calculation Utility (1138039) in Gasoline Direct Injection Engine Simulation in the *Ansys Forte Tutorials*.
- UCX memory allocation error when running Ansys Forte can be overcome by setting an environment variable (1126028). See UCX Memory Allocation Errors in the *Ansys Forte User's Guide*

2. Resolved Issues and Limitations

For a list of Ansys Forte issues that have been resolved in Release 2025 R1, refer to the Resolved Issues and Limitations document on the Ansys Help site.

Chemkin / Energico / Reaction Workbench / MFL Release Notes

The following sections contain release information for Release 2025 R1 of Chemkin products:

1. New Features and Enhancements

Chemkin

- A new Chemkin reactor model, Markstein Calculator, has been added. This model can be used to calculate the Markstein Length, a response of premixed laminar flame to imposed stretch rate.
- Robustness improvements for flame reactor models have been implemented, which allows for computations over a very wide range of strain-rates.
- Licensing updates have been implemented for optimal usage with parameter studies. This update takes advantage of four free license features when running parameter studies.
- New Workflow for Batch ("off-line") Parameter Study runs that consumes Ansys licenses efficiently and enables running batches on a cluster or on several machines. See Running and Post-processing from Batch Scripts in the *Chemkin Advanced Analysis Guide* for details.
- Chemkin-CFD is available on ARM architecture. More details are available on request.
- Added capability of co-simulation of Chemkin Multi-zone HCCI Engine model with GT-SUITE.

2. Resolved Issues and Limitations

For a list of issues that have been resolved in Release 2025 R1, refer to the Resolved Issues and Limitations document on the Ansys Help site.

FENSAP-ICE Release Notes

The following sections contain release information for Ansys FENSAP-ICE 2025 R1.

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 2025 R1, refer to the Resolved Issues and Limitations document on the Ansys Help website.

EnSight Release Notes

The following sections contain release information for Ansys EnSight 2025 R1.

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

Welcome to EnSight 2025 R1!

Help System

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

1. New Features and Enhancements in Ansys EnSight

The following list highlights the enhancements and changes to EnSight in 2025 R1. In addition, we have several major enhancements that have reached Beta state and can be utilized under the guidelines of a beta feature.

- **Improved Visual Attribute Defaults**

  We have updated the default visual attributes in EnSight to take advantage of updates to visual representation in the past few years. The new visual defaults have changed for items such as default background color, default ground plane visualization, default legend palette and legend position, default shadowing, default lights, and default materials. These changes are meant to provide easy application of the many enhancements that had been made in EnSight over the past years. Users who do NOT want the new defaults can start EnSight with the command line option "`-theme 2024 R2`" to force EnSight with the previous default theme. Due to this significant change in visual attributes, 2025 R1 does not automatically copy your preferences from previous version (we leave that up to user to see if they want to pull their preferences on top of the new defaults).

  The theme can be specified in the startup stage (instead of using the version's default) using a command line parameter "`-theme <themename>`" or "`-theme <versionnum>`".

For more details, see Basic Usage in the *Ansys EnSight How-To Manual*.

- **Overlay Hidden Line Improvements**

  For hidden line rendering, EnSight now has a blending weight associated with the hidden line color. Users can now specify a weight to this grid line, allowing you to model surfaces when zoomed out and prevents the surface from being washed out when elements are densely tessellated. As you zoom closer to the surface, the grid lines will become more visible.

- **Part List Filter Widget**

  The Part list now has a filter widget added to the top of the list. This widget can be used to filter the list of parts by name, using a simple filter string which can also include "glob" style wildcard characters.

- **Change to Timeset Handling**

  In previous versions of EnSight, the timeset handling for time values in-between explicitly defined timesteps defaulted to "nearest". For changing geometry transient datasets, this could cause robustness failures in EnSight when time step selections from variable and model geometry timesets did not align due to the default nearest time value handling. To improve robustness and remove ambiguity, when presented with datasets that report to contain time varying geometry connectivity, the time value interpolation handling now defaults to "Left of specified time" for both geometry and associated variable timesets.

- **Omniverse Integration**

  Integration of EnSight with Omniverse continues to develop. Transient solutions are now available, as well as transfer of flipbook scenes and mesh lines from EnSight to Omniverse. We have also utilized native directory storage location, as well as standards based on USD format (open to more than just Omniverse). A User Defined Tool is now also available in EnSight for this export.

- **DVS Improvements**

  We continue development and expansion of the DVS data transfer mechanism. Enhancements in 2025 R1 include performance improvements via Memory Pool implementation, and development towards more flexible caching system.

- **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.

- **Rocky Reader Updates**

  With the first release of the reader in 2024 R2, 2025 R1 continues to expand the reader's capability and scope. Many updates including improvements to time handling, deformation on surfaces, nodeID labels, nodal variables on boundaries, particle release time and residence time variables, neighbor counts, naming conventions and units to name a few.

- **Improved GLTF Export**

The GLTF 2.0 file format is widely adopted in web based applications. Efficient transmission is critical in such situations. EnSight now supports a geometry compression option when exporting GLTF 2.0 files. The compression approach uses Google's Draco compression library. EnSight also provides tessellation resolution control for exporting nodes represented in quadratic shapes.

- **No Longer Copies Preference Files**

  EnSight uses a private preferences directory for each user. A number of different preference files that control EnSight defaults are stored in that directory and are read as EnSight starts up. In previous releases, in a fresh installation of a new release, EnSight always copies the preferences files from its previous release. EnSight stops doing so as of EnSight 2025 R1.

- **Removed Support for the STAR-CD and STAR-CCM+ Reader**

  EnSight no longer includes support for the STAR-CD and STAR-CCM+ Reader. If this reader is required, consider using 2022 R2 or an older release of EnSight that supports this reader.

2. PyEnSight

The following list highlights the enhancements and changes to PyEnSight in 2025 R1.

- **TBD**

  TBD

3. Ansys Dynamic Reporting (Nexus)

See the 2025 R1 version of the Ansys Dynamic Reporting Release Notes for enhancement and change information.

4. Beta in This Release

Documentation for the beta items can be found at Ansys EnSight Beta Manual.

5. Alpha in This Release

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

- **DPF Reader**

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

6. Resolved Issues and Limitations

For a list of issues that have been resolved in Release 2025 R1, refer to the Resolved Issues and Limitations document on the Ansys Help website.

FreeFlow Release Notes

Important:

Ansys FreeFlow™ is a **Beta Software**.

Ansys FreeFlow™

Smoothed-Particle Hydrodynamics (SPH) Simulation Software

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

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

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

See more details about Ansys FreeFlow software on Ansys Website

Learn how-to use with:

- Ansys FreeFlow User Manual
- Ansys FreeFlow Tutorial Guide

Ansys Rocky Release Notes

The following sections contain release information for Ansys Rocky™ particle dynamics simulation software 2025 R1.

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

Welcome to **Ansys Rocky Software** 2025 R1!

1. New Features

In this section you will find all Rocky new features present in the 2025 R1 version.

1.1. General New Features

- **Rocky Solver SDK New Features**

  Native calculations for strain rate tensor and velocity gradient vector was implemented and aims to facilitate the implementation of non-Newtonian fluid modules.

  For more details see Rocky for Developers.

- **PrePost Scripting New Features**

  The support for creating custom curves and custom properties through Rocky scripting tool was added, enabling more automation capabilities to your simulations.

  For more details see Rocky for Developers.

- **Point Clouds in Motion Frames**

  The **Point Cloud** feature enables you to import a text file containing field data that defines one or more points in space. Now with the 2025 R1 version it is possible to add this functionality along with Motion Frames.

  For more details see Rocky User Manual.

1.2. Ansys Rocky Coupling with Ansys Fluent Software

See below the new features you can use to perform the Ansys Rocky coupling with Ansys Fluent® software:

- Advanced option to use the DPM flow blocking approach for a single fluid phase 2-way unresolved coupling.

- 2-equation turbulence models support for 2-way semi-resolved coupling.
- It was added the Fluent Cell volume and cumulative exchanged heat on geometries as new properties related to 2-way coupled problems.
- Volumetric diffusion parameter to control the minimum number of the iterations of the algorithm.
- Advanced option to increase the frequency of update of Fluent mesh coordinates when running 2-way coupled cases with mesh motion.

For more details see Rocky User Manual, and Rocky CFD Manual

2. Improvements

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

2.1. General Improvements

See below the general improvements that this new version brings:

- More detailed Intra-particle collision statistics where information can be drilled down to collisions between each pair of particle groups.
- The input variables support for orientation parameters automization.
- **Volumetric Inlet ennhancements:**
  - To set an initial velocity for particles and SPH elements injected by volumetric inlet.
  - The possibility to inject particles and SPH elements at any time.
- **Transient Point Cloud with time periodic:**
  - Now the time periodic can be used with the Transient Point Cloud, in this way the point cloud field will be repeated for the number of periods you choose.
  - Periodically inject bulks of solid and fluid providind a storage optimization by importing a smaller point cloud.

For more details see Rocky User Manual.

2.2. SPH Improvements

- Perfomance improvements on HTC calculations based on correlations, when the thermal model is disabled.
- Now, when thermal model is enabled, HTC Calculator module calculates HTC values based on fluid adjacent temperature and fluid-triangle heat transfer instead of using correlations, which generates more accurate values.
- Several other improvements related to SPH simulation performance such as the faster neighbour search and a optimized memory layout for SPH element data on GPU.

2.3. Performance Improvements

- The performance of simulations with high triangle count in multi-GPU was improved as well as the 3D rendering of these cases.

2.4. User Experience Improvements

- Data tree elements have been reorganized so that they appear in the order they are generally used when performing a simulation.
- Materials and Materials Interactions were merged to improve usability.
- New **Study** menu as starting point to add Regions of Interest and Point Clouds.
- **Property User Process** was renamed to **Filter User Process.**
- **Frame of Reference** was removed from the Particles Entity as its functionality was misleading the users.
- The ability to extend the periodic motion stop time was added in this release and now when you extend your simulation you can also extend the periodic stop time, without the need of start a new simulation.

For more details see Rocky User Manual.

2.5. Postprocessing Improvements

- Residence time calculations now have the start and stop time functionality.
- Polyhedron User Process can now be attached to motion frames.

For more details see Rocky User Manual.

3. Rocky Modules

In this section you will find new Rocky Modules and main enhancements present in the 2025 R1 version.

Rocky New External Modules 2025 R1

- **EMAG** *(External Module)*

  The Electromagnetic Coupling module enables you to add electromagnetic physics to the simulation, such as:

  – Electrostatic forces for particle-particle and particle-boundary interaction;

  – Charging and discharging effects on particles due contact with boundary surfaces;

  – Field forces on charged or dipole particles immersed on electric/magnetic fields.

  This module replaces the: Electrostatic Force, Magnetic Force and Magnetic Force Ferrous and Tribocharing External Modules.

- **Thornton-Ning Normal Force Module** (External Module)

  This module enables you to use the Thornton-Ning Normal Force for the Normal Forces used in the project. In Rocky software, the Normal Force is used to calculate the normal components of the contact forces, and will therefore affect particle-particle and particle-boundary interactions.

To download the Rocky External Modules, in the Download Center page on Ansys Customer Portal scroll down to **Add-ons Packages**, and in **Ansys Rocky** click on **SDK and Modules** to download all modules available

For more details see Rocky Module Manual.

Rocky New Internal Modules 2025 R1

- **Multibody Dynamic** *(Internal Module)*

  This module replaces the FMU, Ansys Motion Coupling and Adams Coupling External Modules, in this way with just one module you can *import* FMU into Rocky software.

Note:

To do the *export* from Ansys Motion™ Software and Adams you can use the Ansys Motion Coupling and Adams Integration External Modules that continue available in the **Additional Package.**

Note:

The FMU External Module is not available in the 2025 R1, since the **Multibody Dynamic Internal Module** fully replaces this module functionalities for FMU import and export.

Rocky Internal Modules are embedeed in Rocky Software and can be activated in the Modules parameter on Data Panel.

For more details see Rocky User Manual.

4. Beta in This Release

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

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

Important:

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

- **Divergence-Free SPH Incompressible Solver (DFSPH)**

  The new *Divergence-Free Smoothed Particle Hydrodynamics (DFSPH)* brings the IISPH solver improved with a slightly different approach of density correction near boundaries in relation to the one already implemented for IISPH . The main difference is that there is no need of approximating pressure at the virtual ghost particles associated to boundary triangles. This reduces some operations and the need to store a couple of auxiliary arrays.

  For more details see Rocky User Manual.

- **New Pairwise Potential surface tension model**

  A new surface tension model that couple pairwise surface tension model with physical parameters, replacing the *SPH Pairwise Surface Tension* external module.

  For more details see Rocky User Manual.

- **New mapping method for 2-way unresolved coupling with Ansys Fluent software**

  New option on the UI with a new mapping method based on diffusivity equation and back-diffusion.

  The diffusion solution mapping process solves a diffusion equation to perform the mapping and smoothing of the volume fraction variables (particle volume fraction and particle volume fraction variation due to particles entering and leaving the domain) and a pure advection equation to all other quantities. This way all other mapped quantities (averaged particle quantities, momentum, and energy sources, for example) are all distributed according to the volume fraction distribution.

  The Back-Diffusion method uses the same approach as the pure-advection of the DEM quantities applied in the Diffusion Solution mapping scheme, but uses the inverse of the volume fraction diffusion velocity as the advection velocity, to compute a volume fraction weighted transport of the fluid properties and flow quantities at the position where the particles are located.

  For more details see Rocky User Manual.

5. PyRocky

PyRocky highlights and enhancements in 2025 R1 are listed below:

- Allow creation of nested User Processes.
- Expose *RAExportTookit* class.
- Expose *PlaybackMacroFile*function.
- Type Stubs to allow IDEs auto-complete to work with PyRocky objects.

For more details see Rocky for Developers.

6. Rocky Software Installation

6.1. Automated Installer

In this release, Rocky software has easier installation, as it is inside the standard Ansys Automated Installer.

For more details and installation guides see Ansys Automated Installer Guide.

6.2. Rocky Student Version

Following Ansys commitment to setting today's students up for success tomorrow, Ansys Rocky software now has a student version, providing a free simulation software license to students at all levels.

For more details on Rocky Student version limitations see Rocky User Manual.

For more details on Ansys for students see Ansys Student - Free Software Download.

7. Resolved Issues and Limitations

For a list of issues that have been resolved in Release 2025 R1, refer to the Resolved Issues and Limitations document on the Ansys Help website.

Thermal Desktop Release Notes

1. 2025 R1 Service Pack 1 Release Notes

This package contains service pack updates for Ansys products included in the unified installation package. Each service pack release for 2025 R1 will be listed serparately, along with the related issues addressed, within this document.

Ansys Release:

2025 R1

Service Pack:

2025 R1.01

Release Date:

2/12/2025

Issues Addressed (by Product):

Thermal Desktop

2025 R1 1.0 (2/12/2025)

- Ansys Thermal Desktop now supports IFX 2025

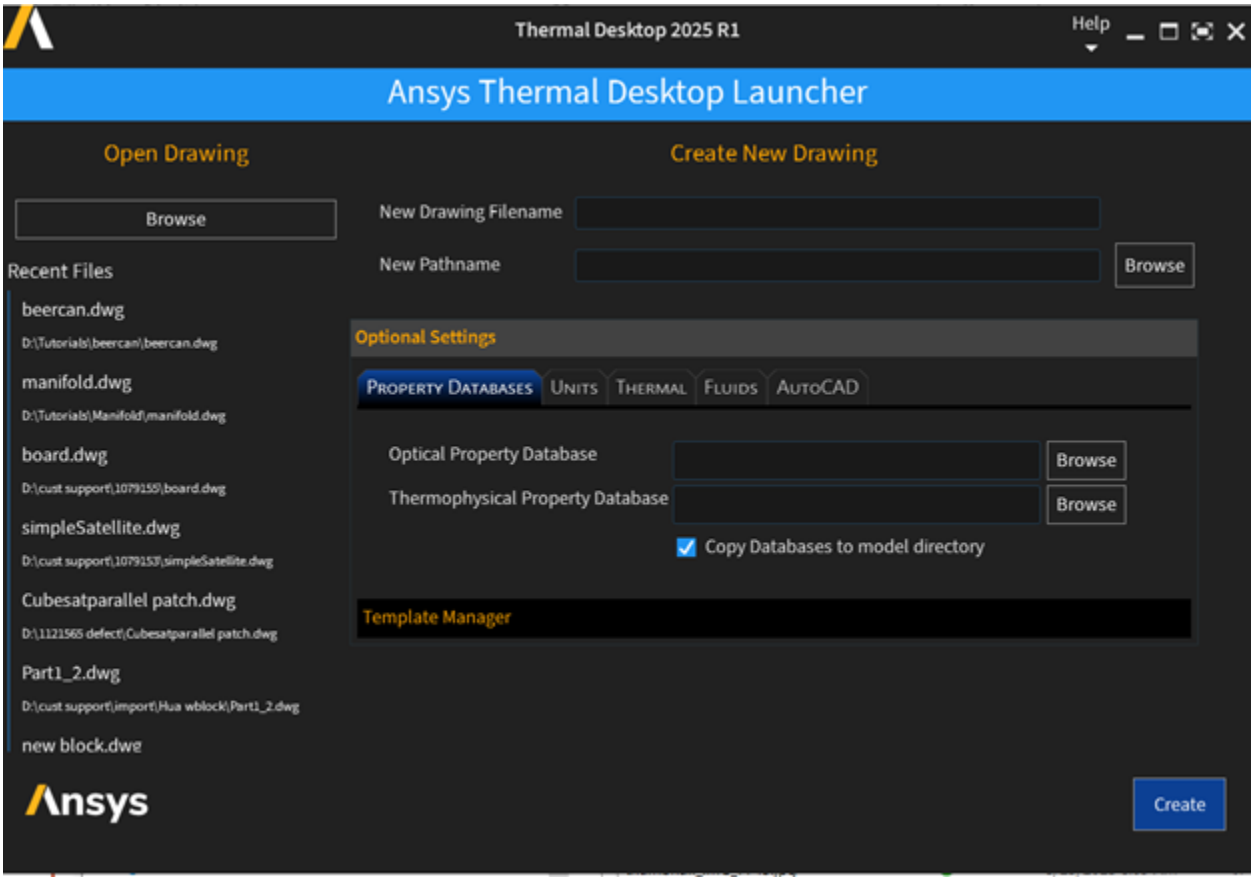
2. 2025 R1 Thermal Desktop Release Notes

2.1. New features

The following changes have been added.

General

- **Thermal Desktop template launcher**- A new Ansys Thermal Desktop application that makes it easy to browse recently opened models, create user templates, and start new models.



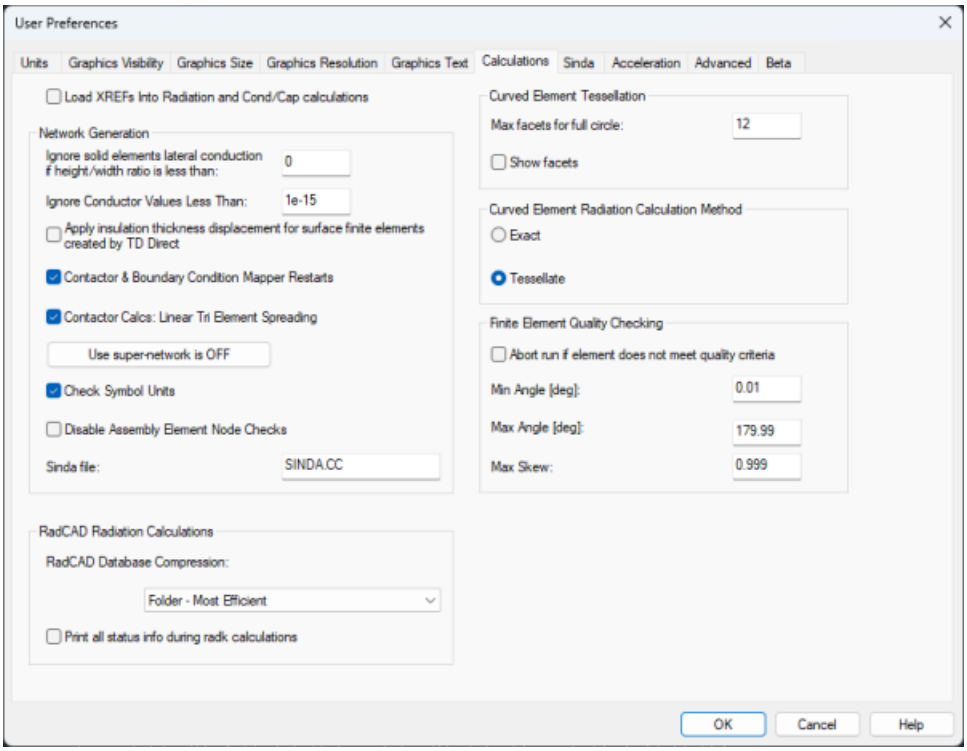
- **User Preferences**- A new check box has been added to User Preferences When checked, evaluation of potential issues with finite elements and articulators is disabled. The potential issues checked are:

  – Duplicate material orienter names. This can cause inconsistent definitions

  – Nodes or mesh manager and related material orienters are not in the same articulator. This can produce unexpected behavior since the material orientation does not move with the elements.

  – All nodes of an element are not in the same articulator. This can cause elements to stretch into poor quality elements.

  Disabling the checks requires you to ensure the model is valid in all possible orientations of the articulators.



- **Enhanced Restarts**- Phase Change Material (PCM), recession and accretion data can be initialized in a case from a previous save file. Other arrays can also be initialized from a previous save file.
- Speed improvements made for compiling, initializing and executing PCM, recession and accretion models. Cases that used to take 30+ minutes to compile because of large amounts of generated lines of code now take only seconds.
- Mass and energy errors for lumps now can be saved in a User Defined Fortran Array (UDFA) for post processing.
- Access to Ansys Online Help.
- Integration of Ansys Thermal Desktop to Ansys Model Center.

Thermal Modeling

- **Enhanced Restarts**- Previously restarts were not possible for certain cases. Phase Change Material (PCM), recession and accretion data can be initialized in a case from a previous save file. Other arrays can also be initialized from a previous save file.
- Sinda submodels quantity updated to 9999, previously the limit was 999.

Radiation Modeling

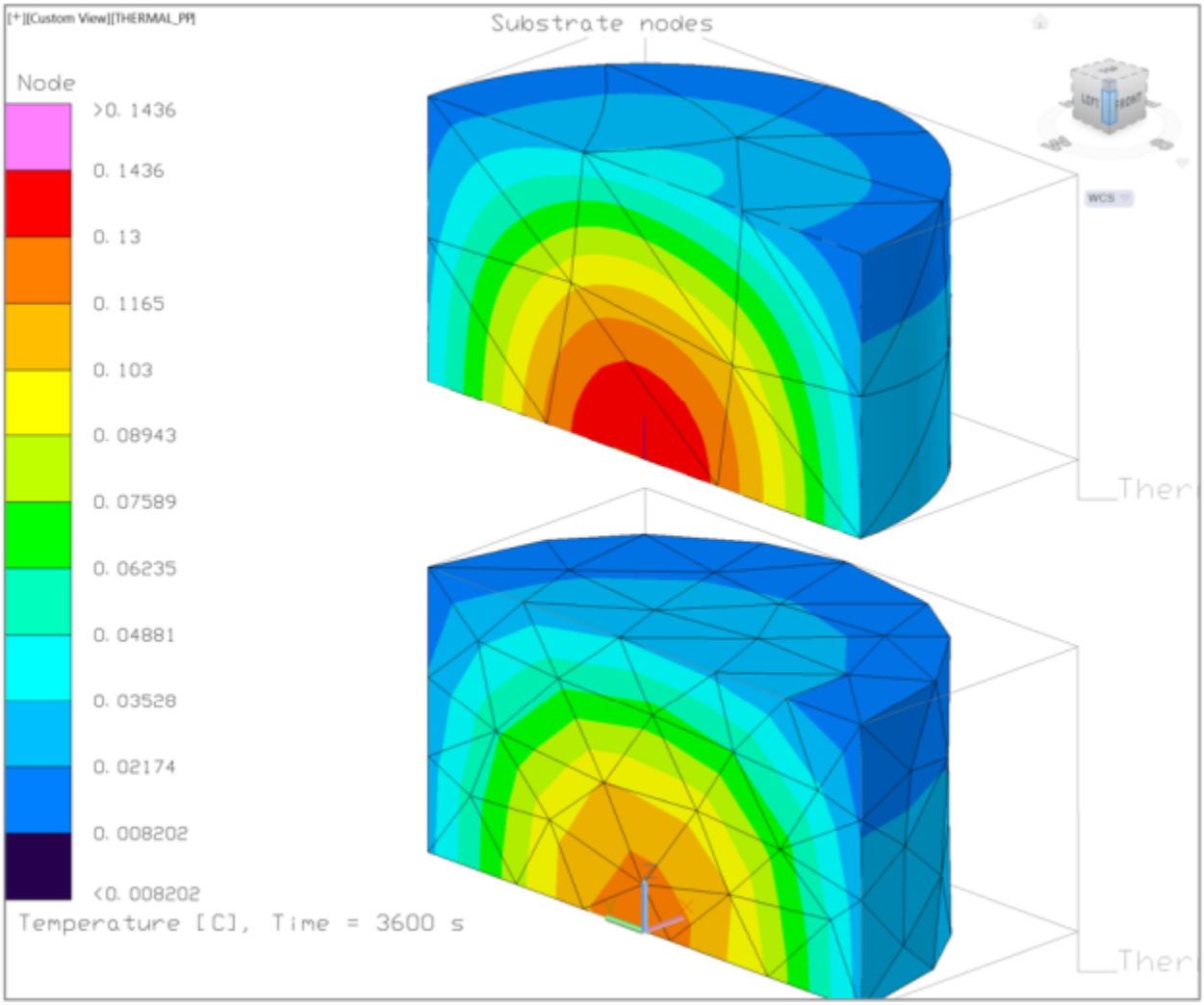
- N/A

Fluids Modeling

- Option to exclude Plenum from Lump initializations in the Case Set Manager.
- Volumetric Heat Loads can be applied to RcPipe walls can now be used to model wiring with Joule/Ohmic heating.

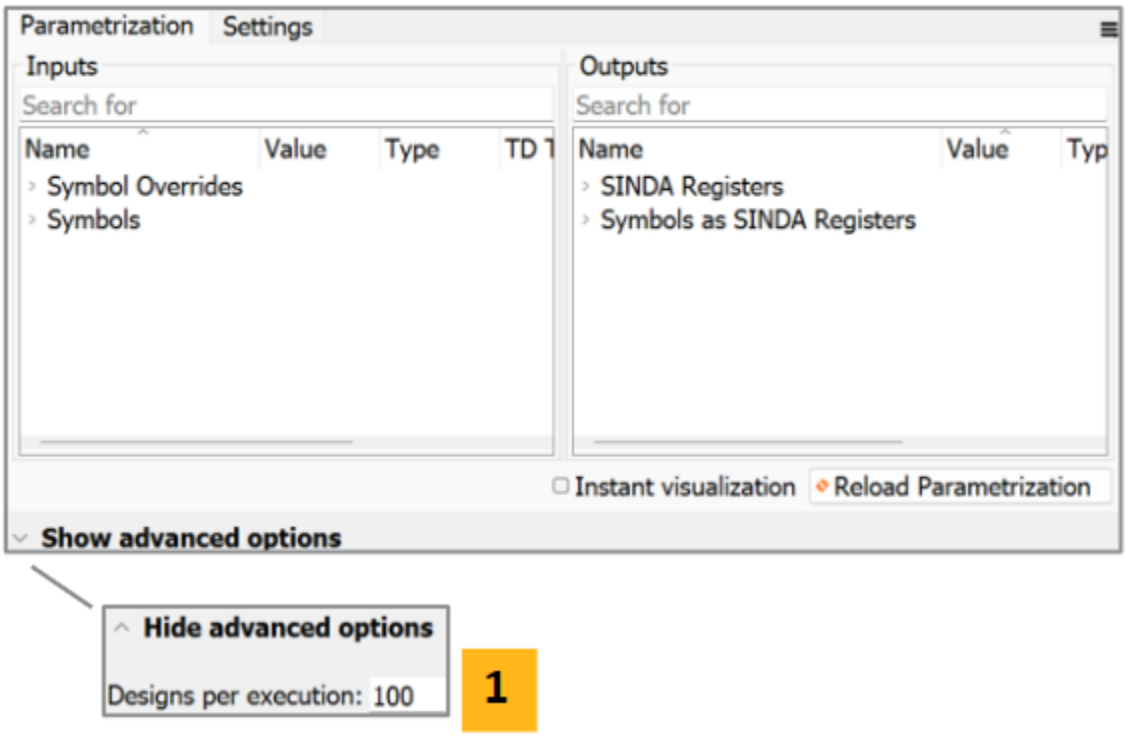
2.2. Beta Features

- Second order finite elements (curved thermal elements with higher order thermal solutions). May yield either accuracy and/or speed improvements in some models.

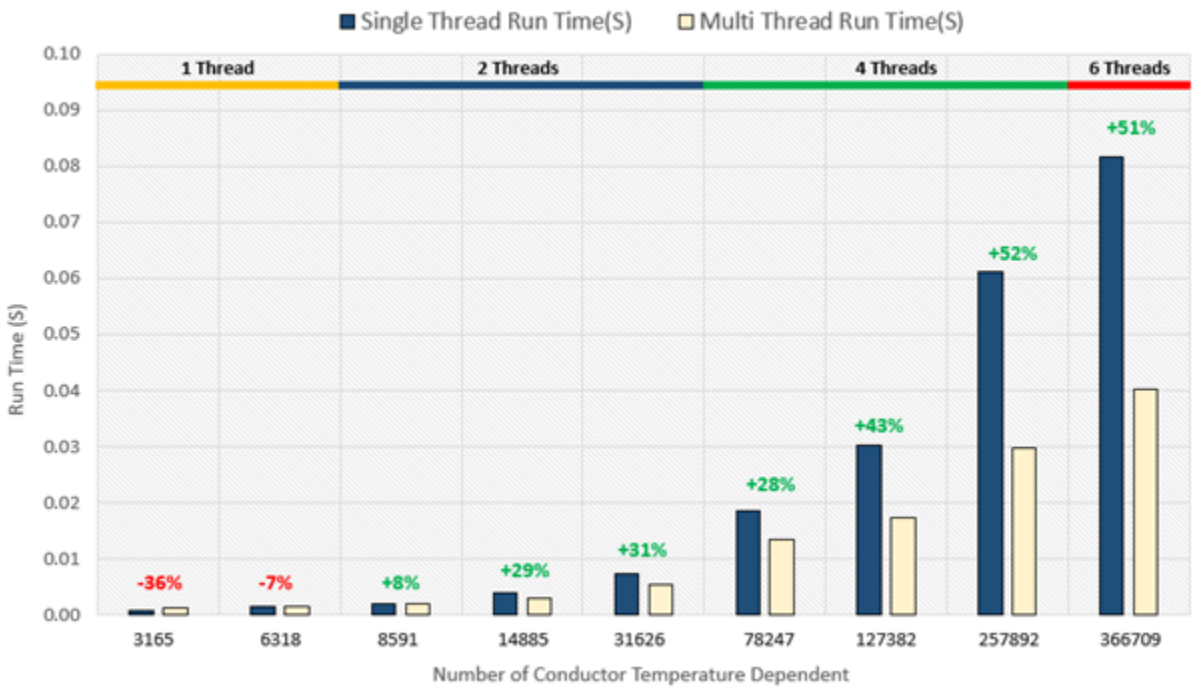


- Not yet compatible with Compartment walls.
- **Ties in solids**- Intended for small fluid passages where accuracy in convection is secondary, and where not disrupting solid meshes to account for such passages is primary.
- **Participating media**- For thick glass optics, or for treating optically thick simple gas flows. Currently restricted to thermal solids.

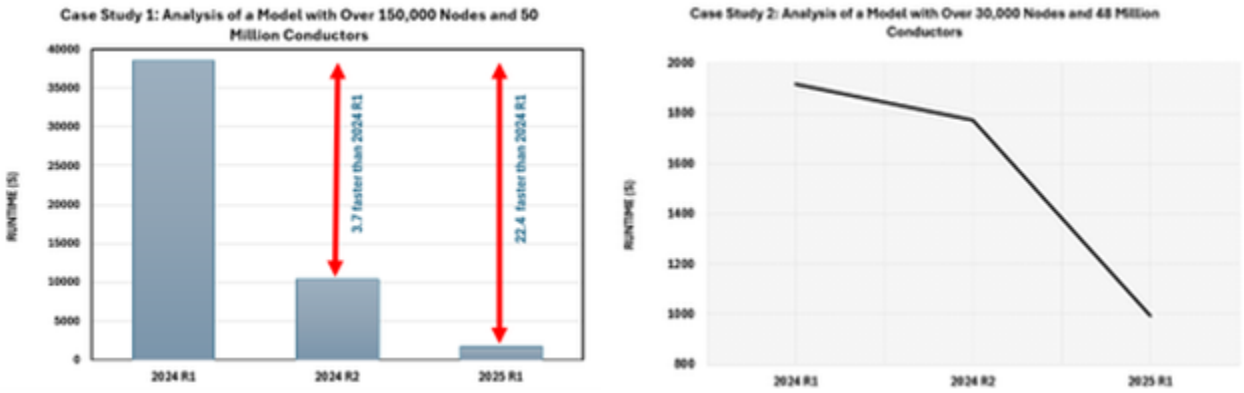
- New binary format (SaveX) for increased speed and reduced file sizes. This will replace both SAVE and CSR formats when mature.
- Interface to Ansys OptiSlang:



- Multi-thread run time improvements:



- Speed improvements for Time and Temperature Dependent Calculations
- **Sinda enhanced Sparsification**- resulting in 4x speed improvements in some test cases:



2.3. Deleted Features

Meshing in Thermal Desktop (TD Mesher) has been removed, TD Direct which is included with Ansys Thermal Desktop has better meshing capabilities and users should migrate to using TD Direct for meshing. Models that currently have mesher created with TD Mesher will still be valid, but new meshing is not possible.

Critical Bug report:

Description of Error:

The Dyer's method (MCH=-3) is the default for predicting metastable two-phase critical flow rates in the throats of constrictions. It is principally intended for two-phase inlets, or for volatile liquid inlets flashing into the dome. But the theory is good for superheated vapor at the inlet that drops into the dome (two phase throat).

Unfortunately, the algorithm wasn't perfectly symmetric, and had to be revised when the downstream pressures are very low and supercritical vapor is upstream.

These are the combination of factors where this can happen:

- Condensable vapor present in the inlet.
- XL=1.0 (dry) inlet.
- Inlet at high pressures (not necessarily supercritical pressures, but that is most likely since superheated but subcritical inlets aren't often at high enough pressures to see this effect).
- Outlet at low pressures (low enough to go into the dome when overexpanded to the downstream pressure ... otherwise the algorithm would have correctly kicked out of Dyer’s in the prior version).

First Incorrect Version: 2023 R2 (23.2)

Corrected in: 2025 R1

Suggested User Action for Running on Uncorrected Version:

Use MCH=-2 for superheated vapor upstream of constrictions (AFTH smaller than AF) such as orifices and inlets, instead of letting the program switch automatically (to MCH_USED = 2P or -2P) from the default of MCH=-3.

Ansys Electronics Products

Release notes are available for the following Ansys Electronics products:

Icepak Release Notes

Release 2025 R1 of the Ansys Icepak application offers most of the capabilities from previous releases plus an enhancement.

1. Introduction

Ansys Icepak 2025 R1 is a release of Ansys Icepak that has a new feature and resolved issues and limitations.

2. New and Modified Features in Ansys Icepak 2025 R1

Release 2025 R1 of Icepak has no new features or enhancements.

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

3. Resolved Issues and Limitations in Ansys Icepak 2025 R1

For a list of issues that have been resolved in Release 2025 R1, 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:

Speos Release Notes

The following sections present the new features, enhancements and changes for Speos 2025 R1 release.

1. New Features

This section describes the new features introduced in Speos 2025 R1 release.

Features from Beta to Official

- **Physical Camera**: the Physical Camera is now officially released and no longer in BETA mode. The Physical Camera allows you to create a lens system with its own optimized propagation engine, to be included in a bigger optical system. The Physical Camera sensor will improve the convergence of the rays to provide more accurate results.

Speos Libraries - PySpeos

- **PyAnsys** is a collection of **open-source Python libraries** designed for engineers to enhance the capabilities of Ansys products. As of version 2025 R1, **PySpeos** will be publicly available on GitHub and include sample codes and workflows.
  - PySpeos provides a direct access to Speos solver without considering Speos graphical user interface or CAD software - which was required until now when scripting in Speos.
  - PySpeos enables to automate core level actions such as creating new features or modifying existing ones, and complex workflows.

Materials

- As of version 2025 R1, Speos allows you to use AxF files as material properties as it stores many effects through multiple texture maps. AxF (Appearance eXchange Format) is a textured appearance property file generated with X-Rites measurement devices, and wildely used in Automotive for rendering purpose.

Component

- **3D Texture**: Sometimes, the mapping file may not be enough to manufacture the result of a 3D Texture. As of version 2025 R1, you can export your 3D Texture created in Speos as geometry in a *.stl file to allow you to continue the manufacturing process. Indeed, the *.stl file can be used in addition to the mapping file that describes the different positions, orientations and size variations of the replicated patterns across the support geometry. Thus, replicated patterns and support geometry can be exported as two distinct *.stl files. Speos can also export the geometries corresponding to the result of the Boolean operation as *.stl file.

Live Preview

- Multiple improvements to enhance analysis and navigation in the Live Preview
  - New Result Legend tool (Color scale) available providing you with different capabilities to customize the representation of the result and the data interpretation.

  - New Measure Picker tool to display the photometric/radiometric value at the position of the mouse cursor.

  - New Photometry/Radiometry switch to switch the Color Scale, Measure Picker and Light Controller units into photometric (lm based) or radiometric (W based)

  - New navigation tool: Set Spin Center allows you to define a new rotation center in third person navigation around which the view will rotate.
  - New Linear pixel scaling parameter to activate the color smoothness between neighbor pixels (only for visualization).
  - New Pin tabs at opening parameter to holds still the Sensors and Lights panels at next opening.
  - The first and third person 3D navigation are now up-preserved: the camera tilt will no longer tilt sideway and will maintain its initial up orientation. The navigation in Immersive Speos360 results is also up-preserved.

Optical Part Design

- **Optical Lens**: the Optical Lens feature now offers the possibility to create TIR Lenses directly from its definition panel and merge these TIR Lenses with the Optical Lens. The goal is to ease the creation of optical components combining Optical Lenses and TIR Lenses.

- **Optical Lens/Optical Surface**: New option Custom contour available. Custom contour allows you to select a surface body that will be used as a pattern to trim the Optical Part Design feature. This will give the feature a particular shape created out of the surface body contour.

2. Enhancements

This section describes the enhancements introduced in Ansys Speos 2025 R1 release.

Meshing

- Speos simulation relies on the meshing that is extracted from the selected geometries. However, setting the meshing parameter to get an accurate representation of the geometries is not easy. In addition, some combinations of meshing parameters may lead to non-watertight result causing possible propagation error.

  Therefore, as of version 2025R1, new default meshing parameters values are applied upon creation of the following features, in order to provide a more relevant and simplified default meshing than prior versions.

  – Interactive, Direct, Inverse, LiDAR simulations

  – Light Box Export

  – Local Meshing

- Meshing improvement regarding the Angle meshing setting: now you can set a 1° minimum angle. Before version 2025 R1, the minimum threshold was 12°.

Component

- **Speos Pattern**: Speos Patterns used to created multiple instances of a same source are now compatible with Interactive simulations.
- **3D Texture**: Better management of 3D texture patterns construction on edges of support geometries.

Sensor

- **Physical Camera**: Physical Camera sensors are now compatible with black-boxed Speos Light Boxes. Therefore, along with the encrypted Speos Light Box to restricts the access and protect the intellectual property:

  – No details about 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.
  - the *.OPTSequence file generated and used by the Physical Camera Sensor is encrypted.

- **Physical Camera**: As technology evolves along with space constraints, lens systems such as periscope ones are introduced into new products. Physical Camera Sensor has been designed originally for any type of on-axis lens systems.

  As of version 2025 R1, you can benefit from the fast calculation of stray light contributions on off-axis lens systems.

- **Camera sensor**: algorithm has been improved to avoid lag time when using a significant *.OPTDistortion Binary file.
  - Before, the Camera sensor was automatically computed every time a modification was done to the *.OPTDistortion Binary file.
  - Now, the Camera sensor needs to be manually computed to take into account the modification. If the Camera sensor is not manually computed, then simulations take the latest parameters into account.

Simulation

- The GPU algorithm has been improved to generate more adequate results regarding the luminance analysis.
- The GPU algorithm has been improved for a better grazing ray detection on GPU results.
- The GPU algorithm has been improved to better manage the total internal reflection (TIR) for unpolished surfaces in simulation.

Optical Part Design

- **Light Guide**:
  - Light Guide improvement regarding construction and migration to prevent light leakage at first prism and last prism of the guide.

  - Light Guide improvement: Width parameter is now compatible with the **Control points** and **Input files** mode in **Hybrid** operation.

  - Light Guide algorithm has been improved to allow the use of significant curvature of the guide curve.

3. Changes

This section describes the operational and support changes introduced in Ansys Speos 2025 R1 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 **553.09** is required from 2025 R1.

4. Resolved Issues and Limitations

For a list of issues that have been resolved in Release 2025 R1, refer to the Resolved Issues and Limitations document on the Ansys Help website.

Speos for NX Release Notes

The following sections present the new features, enhancements and changes for Speos for NX 2025 R1 release.

1. New Features and Enhancements

This section describes the new features and enhancements introduced in Speos for NX 2025 R1 release.

Speos Libraries - PySpeos

- **PyAnsys** is a collection of **open-source Python libraries** designed for engineers to enhance the capabilities of Ansys products. As of version 2025 R1, **PySpeos** will be publicly available on GitHub and include sample codes and workflows.
  - PySpeos provides a direct access to Speos solver without considering Speos graphical user interface or CAD software - which was required until now when scripting in Speos.
  - PySpeos enables to automate core level actions such as creating new features or modifying existing ones, and complex workflows.

Source/Sensor

- New **Light Field** feature available: The Light Field Source and Sensor allow you to create a propagation result of a sub-optical system to be reused in a more complex optical system to gain time when computing simulation.

  The Light Field feature is in BETA mode for the current release.

Sensor

- New **Physical Camera sensor** available: The Physical Camera has been designed to speed up simulation time when it comes to decreasing statistical noise on all the possible optical paths inside the optical system. Simulations with a Physical Camera sensors are faster when considering all the optical paths:

  - Rays are aimed toward the entrance of the optical system, which improves the probability for rays to enter the optical system, in direct propagation.
  - Simulations are also faster, considering specular optical paths found during a pre simulation. User can focus on the sequences of interest.

Simulation

- The GPU algorithm has been improved to generate more adequate results regarding the luminance analysis.
- The GPU algorithm has been improved for a better grazing ray detection on GPU results.
- The GPU algorithm has been improved to better manage the total internal reflection (TIR) for unpolished surfaces in simulation.

Live Preview

- Multiple improvements to enhance analysis and navigation in the Live Preview
  - New Result Legend tool (Color scale) available providing you with different capabilities to customize the representation of the result and the data interpretation.

  - New Measure Picker tool to display the photometric/radiometric value at the position of the mouse cursor.

  - New Photometry/Radiometry switch to switch the Color Scale, Measure Picker and Light Controller units into photometric (lm based) or radiometric (W based)

  - New navigation tool: Set Spin Center allows you to define a new rotation center in third person navigation around which the view will rotate.
  - New Linear pixel scaling parameter to activate the color smoothness between neighbor pixels (only for visualization).
  - New Pin tabs at opening parameter to holds still the Sensors and Lights panels at next opening.
  - The first and third person 3D navigation are now up-preserved: the camera tilt will no longer tilt sideway and will maintain its initial up orientation. The navigation in Immersive Speos360 results is also up-preserved.

Meshing

- Speos simulation relies on the meshing that is extracted from the selected geometries. However, setting the meshing parameter to get an accurate representation of the geometries is not easy. In addition, some combinations of meshing parameters may lead to non-watertight result causing possible propagation error.

  Therefore, as of version 2025R1, new default meshing parameters values are applied upon creation of the following features, in order to provide a more relevant default and simplified meshing than prior versions.

  – Interactive, Direct, Inverse, LiDAR, Virtual BSDF Bench simulations

  – Light Box Export

  – Local Meshing

- Meshing improvement regarding the Angle meshing setting: now you can set a 1° minimum angle.

Optical Part Design

- The **Photometry tool** is now accessible in the Feature Viewer for Optical Surface, Optical Lens, and Honeycomb Lens. The Photometry tool generates a photometric preview displayed in the feature viewer, along with a HTML report and a XMP file for the selected elements. This helps you verify whether your current Optical Surface or Optical Lens design meets the defined regulatory standards.

- **Light Guide** improvement regarding construction and migration to prevent light leakage at first prism and last prism of the guide.

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 2025 R1 release.

Versions Update

- Windows: Windows 10 and Windows 11

  Minimum NVIDIA Driver **553.09** is required from 2025 R1.

Operating System

- NX 2212 up to NX 2212 (2212.9140) service pack is supported in 2025 R1.
- NX 2306 up to NX 2306 (2306.9120) service pack is now supported from 2025 R1.
- NX 2312 up to NX 2312 (2312.8500) service pack is now supported from 2025 R1.
- NX 2406 is no longer in BETA and now officially released from 2025 R1.

  NX 2406 up to NX 2406 (2406.4000) service pack is now supported from 2025 R1.
- NX 2412 is in BETA mode as of version 2025 R1.

3. Resolved Issues and Limitations

For a list of issues that have been resolved in Release 2025 R1, refer to the Resolved Issues and Limitations document on the Ansys Help website.

Speos for Creo Parametric Release Notes

The following sections present the new features, enhancements and changes for Speos for Creo Parametric 2025 R1 release.

1. New Features and Enhancements

This section describes the new features and enhancements introduced in Speos for Creo Parametric 2025 R1 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 2025 R1 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.

Deprecated and End of Life Features

- The Deterministic algorithm with and without Photon Map is **deprecated as of version 2024 R2** and will be in **end of support in version 2025 R1**.
- The Camera Sensor in Geometric Mode is **deprecated as of version 2024 R2** and will be in **end of support in version 2025 R1**.
- The Camera Post Processing is **deprecated as of version 2024 R2** and will be in **end of support in version 2025 R1**

3. Resolved Issues and Limitations

For a list of issues that have been resolved in Release 2025 R1, refer to the Resolved Issues and Limitations document on the Ansys Help website.

Speos Labs Release Notes

The following sections present the new features and changes for Speos Labs 2025 R1 release.

1. New Features and Enhancements

This section describes the new features and enhancements introduced in Speos Labs 2025 R1 release.

Virtual Photometric Lab/Virtual Human Vision Lab

- With the new option introduced in version 2024 R2 **Save Display Options as default**, you save a XML template in the %appdata% folder, that you can quickly apply using the other new option **Load Display default**.

  As of version 2025 R1, this XML template saved in the %appdata% folder is now automatically applied on the XMP result when you run a simulation, so that you do not have to apply it manually with the option **Load Display default**.

2. Changes

This section describes the operational and support changes introduced in Ansys Optical Labs 2025 R1 release.

- Windows: Windows 10 and Windows 11

  Minimum NVIDIA Driver **553.09** is required from 2025 R1.

3. Resolved Issues and Limitations

For a list of issues that have been resolved in Release 2025 R1, refer to the Resolved Issues and Limitations document on the Ansys Help website.

Speos HPC Release Notes

The following sections present the changes and enhancements for Speos HPC 2025 R1 release.

1. New Features and Enhancements

Operating System

- Speos HPC now supports the Linux version:
  - **Rocky Linux 8.10**
  - **Rocky Linux 9.4**

GPU

- Windows: Windows 10 and Windows 11

  Minimum NVIDIA Driver **553.09** is required from 2025 R1.

- Linux

  Minimum NVIDIA Driver **550.107.02** is required from 2025 R1.

2. Resolved Issues and Limitations

For a list of issues that have been resolved in Release 2025 R1, 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 Release Notes

The following sections present the new features, enhancements and changes for Ansys Sound™ products in the 2025 R1 release.

1. Ansys Sound: Analysis and Specification

The following sections present the new features and enhancements for the Ansys Sound: Analysis and Specification™ application 2025 R1 release.

1.1. New Features

Sound Power Level in Watts

Functionality has been introduced to allow calculation of Sound Power Level in watts (SWL) according to ISO 3744.

PyAnsys Sound

Sound Power Level and Power Spectral Density Calculation features added for scripting in PyAnsys Sound.

Graphs and Colormaps

it is now possible to display the position of Cursors directly on the graph for time and frequency windows and to display the position of Time and Frequency slices directly on a spectrogram colormap.

1.2. Enhancements

Revised Sound Generation Methods from a Spectrum

Sound generation methods using a Spectrum input have been reorganized to clarify the and streamline the process.

User Interface Update

Icons in the UI have been updated to align with Ansys Design Language guidelines.

Installation

Sound applications are now included in the Ansys Automated Installer.

1.3. Resolved Issues and Limitations

For a list of issues that have been resolved in Release 2025 R1, refer to the Resolved Issues and Limitations document on the Ansys Help website.

2. Ansys Sound: ASDforEV

The following sections present the new features and enhancements for the Ansys Sound: ASDforEV™ application 2025 R1 release.

2.1. Resolved Issues and Limitations

For a list of issues that have been resolved in Release 2025 R1, refer to the Resolved Issues and Limitations document on the Ansys Help website.

3. Ansys Sound: Jury Listening Test

The following sections present the new features and enhancements for the Ansys Sound: Jury Listening Test™ application 2025 R1 release.

3.1. Enhancements

Audio Clipping and Calibrated Playback

Documentation for Audio Settings has been enhanced and a troubleshooting section on Audio Clipping has been added to help users resolve issues with clipping during sound playback.

3.2. Resolved Issues and Limitations

For a list of issues that have been resolved in Release 2025 R1, refer to the Resolved Issues and Limitations document on the Ansys Help website.

4. Ansys Sound: VR Sound

The following sections present the new features and enhancements for the Ansys Sound: VR Sound™ application 2025 R1 release.

4.1. New Features

TCP/IP Support for API

The Ansys Sound: VR Sound API can now be driven using the OSC protocol over TCP/IP as well as UDP.

4.2. Resolved Issues and Limitations

For a list of issues that have been resolved in Release 2025 R1, 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:

AVxcelerate Lighting and Sensors Release Notes

The following section presents the enhancements and changes for AVxcelerate Lighting and Sensors 2025 R1 release.

1. Enhancements

This section describes the enhancements introduced in AVxcelerate Lighting and Sensors 2025 R1 release.

1.1. Co-simulation with CarMaker 13

You can now perform Co-simulation with CarMaker version 13.0. 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 13 version.

AVxcelerate Lighting and Sensors is compatible with CarMaker versions 11.0, 12.0.1 and 13.0. CarMaker 10 is no longer supported.

1.2. Static Assets and Traffic Signs added to the Library for CarMaker

Thanks to the addition of numerous static assets and traffic sign to the Library for CarMaker, you can now customize CarMaker Roads.

2. Change

This section describes the change introduced in AVxcelerate Lighting and Sensors 2025 R1 release.

2.1. Product Installation Through the Ansys Automated Installer

To install AVxcelerate Lighting and Sensors, as well as its related libraries, for SCANeR and for CarMaker, you should now use the Ansys Automated Installer available from the Current Release page of the Ansys Customer Portal.

Important notes

- Select the **AVxcelerate Headlamp** product to install AVxcelerate Lighting and Sensors and the **AVxcelerate Headlamp Library for SCANeR** or **AVxcelerate Headlamp Library for CarMaker** according to the driving simulator you use.
- Uninstalling any older version of the software is no longer required before installing the new version. However, if you install the 2025 R1 version without uninstalling a previous version,

please be aware that the files that were installed along with the software in *C:/Program-Data/Optis* will be overwritten with the newer version.

- The files that were previously installed when selecting **AVxcelerate Lighting and Sensors Cosimulation Library** are now installed when selecting the **AVxcelerate Headlamp** product.

The Virtual Regulation plugins for AVxcelerate Lighting and Sensors (IIHS, CIASI and FMVSS108) are now available for download from the **Add-On Packages** section of the Current Release page.

AVxcelerate Sensors Release Notes

The following sections present the new features, enhancements and changes for AVxcelerate Sensors 2025 R1 release.

1. New Features

This section describes the new features introduced in AVxcelerate Sensors Simulator 2025 R1 release.

1.1. Radar Simulation: Propagation Output

You can now obtain information on the electromagnetic propagation of a radar with respect to the dielectric properties of materials in the scene, thanks to the simulation mode for propagation output. Available as a beta feature in the 2025 R1 release of AVxcelerate Sensors Simulator, you can now use the Sensor Modeler application or the Sensors REST API to create a radar model with a mode which produces propagation output. Simulate the electromagnetic propagation by adding the radar to a sensor configuration and setting the relevant radar simulation parameters.

For more details on the generated output, refer to Propagation Output Data in the AVxcelerate Sensors Simulator User's Guide.

1.2. IP Protection for Camera Sensors

In the 2025 R1 release, it is now possible to extend IP protection to your cameras and fisheye cameras. To maintain the confidentiality of your sensor component details, from the Sensor Modeler or Sensors REST API, you can now activate protection on the **Imager** section of a camera or fisheye camera. IP protection for camera sensors allows for:

- a first layer of encryption for **.camerax* files which prevents its data from being read without the use of AVxcelerate Sensors applications.
- a second layer of encryption for the **Imager** section of a camera or fisheye camera model and its associated files thanks to a password protection feature in the Sensor Modeler and the Sensors REST API.

When Protection is activated on the **Imager** section of a camera or fisheye camera, only the **Native Resolution** parameters from this section can be viewed or modified.

2. Enhancements

This section describes the enhancements introduced in AVxcelerate Sensors Simulator 2025 R1 release.

2.1. AVxcelerate Sensor Labs User Interface Enhancements

In 2025 R1 the AVxcelerate Sensor Labs software has renewed its focus on the user experience. Find a much-improved version of the Sensor Modeler and Sensor Layout applications which not only benefit from an overhaul of the interface for a more visually appealing experience, but from new functionalities for a better navigation of your libraries. Among the enhancements you can find:

- a **Change theme** button to switch between light mode and dark mode
- a redesign of the sensor and sensor layout cards
- the addition of a **List View** for your libraries
- information regarding your sensor's status and the content of your sensor layouts
- filters for refined searches
- the ability to download or delete using multiple selection.

For more details, refer to the Sensor Modeler and Sensor Layout sections of the AVxcelerate Sensors Simulator User's Guide.

2.2. Corner Reflector Samples

Some samples that demonstrate the use of corner reflectors to calibrate radar sensors are now delivered in the standalone and co-simulation libraries.

In the AVxcelerate Standalone Library, the sample is a python script located in the *%AVXLIB_ROOT251%\Samples\CornerReflectors* folder. For more details, refer to the *README.md* file.

In the AVxcelerate SCANeR Sensors Cosimulation library, three sample scenarios are provided in your *SCANeRstudio_2021* folder in *data\VRXPERIENCE_SENSORS_2025R1\scenario\CornerReflectorsScenarios*. For more details on each scenario, refer to the scenario's description.

In the AVxcelerate CarMaker Sensors Cosimulation library, three sample TestRuns are provided in your *CM_Projects* folder in *AVX_SENSORS\2025R1\DataPools\CM[version]\Data\TestRun\CornerReflectorsScenarios*. For more details, refer to the *README.txt* file.

2.3. Co-simulation with CarMaker 13

AVxcelerate Sensors Simulator is now also compatible with CarMaker version 13.0 and CarMaker HiL version 13.0. CarMaker version 10 is, however, no longer supported.

The CarMaker and CarMaker HiL versions supported by AVxcelerate Sensors Simulator 2025 R1 are 11.0, 11.1, 12.0, 12.0.1, 12.0.2, 13.0.

2.4. New Import Feature Added to Asset preparation API

The AVxcelerate Asset preparation API v3 delivered in 2025 R1 allows you to import assets and tracks to the server thanks to new methods added to the *Environment Preparation* and *Scene Tree Preparation* services.

For more details on this new feature and about other changes introduced in this release, refer to the changelog provided in the API documentation.

2.5. Sensors Simulation Improvements

The following improvements have been made to sensor simulation:

- Simulation performance across all sensors has improved thanks to the optimization of data serialization resulting in the reduction of periodic slowdowns.
- A black image is displayed when incompatible output formats are configured and **Display** is enabled in the camera simulation parameters.
- It is now possible to define a delay for a sensor simulation using the **start offset** parameter. This parameter, which can be applied to any sensor type, can be used:
  - to optimize and better control the frequency and consistency of sensor data production. Refer to Update Frequency and Sensor Production Period in the AVxcelerate Sensors Simulator User's Guide to understand the impact of this parameter on data production.
  - to simulate a multi-radar configuration designed to reduce interference among radars. Set the **start offset** in the radar simulation parameters.
  - to simulate a High Dynamic Range (HDR) camera through a multi-camera configuration. Refer to the **start offset** parameter in the camera simulation parameters to understand how to use this parameter to simulate an HDR camera.

2.6. New Animated Pedestrian for LiDAR and radar Simulations

The new *pedestrian_male_lidar_radar.asset* file provided in the AVxcelerate Sensors Standalone Library is an animated pedestrian designed for LiDAR and radar simulations. The corresponding CarMaker object is provided in the CarMaker AVxcelerate Sensors Library for CarMaker.

Note:

This file has been adapted from an Adobe Mixamo animated character (https://www.mixamo.com/). As a consequence, you must comply with the Adobe terms of use (Adobe General Terms of Use) when using these specific assets. In particular, you must not use those assets "to directly or indirectly create, train, test, or otherwise improve any machine learning algorithms or artificial intelligence system, including but not limited to any architectures, models, or weights" (extract from section 17. of Adobe terms of use).

---

3. Changes

This section describes the changes introduced in AVxcelerate Sensors 2025 R1 release.

3.1. Products Installation Through the Ansys Automated Installer

To install AVxcelerate Sensors, as well as its related libraries, for SCANeR and for CarMaker, you should now use the Ansys Automated Installer available from the Current Release page of the Ansys Customer Portal.

Important notes

- When you install AVxcelerate Sensors on a Linux OS workstation, you can now select the installation directory.

  The default installation directory path for installing the product on Linux OS is now */usr/ansys_inc*, which differs from the installation directory of previous versions (which was */ansys_inc/*)

- The set of files that were previously installed when selecting **AVxcelerate Standalone Library** are now installed along with the **AVxcelerate Sensors** product. You can still select the installation directory for this library, which is now named **AVxcelerate Sensors Standalone Library** during the installation.

3.2. Remote Direct Memory Access

Remote Direct Memory Access (RDMA) is now available in the 2025 R1 release of AVxcelerate Sensors Simulator as a beta feature using NI RDMA technology. Activated through the camera simulation parameters, the RDMA component allows for the transfer of camera and imager output to an RDMA receiver via the NI RDMA card. Previously released as an experimental feature, in the 2025 R1 release:

- the **Buffer Size** parameter has been renamed **Transfer Chunk Size**

- four additional parameters have been added to allow you to add pre-embedded and post-embedded data lines to the transferred data.

In the 2025 R1 release of AVxcelerate Sensors Simulator, the RDMA plug-in is not provided in the installer. If you wish to use RDMA to transfer your camera simulation data, you must request the RDMA archive and pre-requisite files.

3.3. Deprecation of Sensors created with version prior to 2022 R2

AVxcelerate Sensors Simulator, AVxcelerate Sensor Labs (applications and REST APIs), as well as the unpacker executable no longer support the sensors created with a version prior to 2022 R2.

3.4. VSS API Changelog

This section lists all the changes introduced in VSS API version 2025 R1 compared to version 2024 R2.

3.4.1. Changes in *simulation_parameters.proto*

- The *start_offset* field has been added to the **SensorParameters** message.
- In the **RemoteDirectMemoryAccess** message:
  - the *buffer_size* field has been renamed *transfer_chunk_size*, but the index of this field has not changed.
  - the following field have been added:
    - → *post_embedded_data_line_count*
    - → *post_embedded_data_line_payload_size*
    - → *pre_embedded_data_line_count*
    - → *pre_embedded_data_line_payload_size*

3.4.2. Changes in *lighting_system_state.proto*

In the *lighting_system_state.proto* file, the *light_functions* field has been added to the **LampState** message.

Ansys Mesh Prep Products

Release notes are available for the following Ansys Mesh Prep products:

Meshing Application Release Notes

Release 2025 R1 of the Meshing application contains many new features and enhancements. Areas where you will find changes and new capabilities include the following:

Many of the enhancements detailed in the Mechanical Application Release Notes are also relevant to the Meshing application.

1. Changes in Product Behavior from Previous Releases

The **2025 R1** release includes the following changes in product behavior:

- **Mesh Workflow Worksheet** is renamed to **Property Worksheet**.

2. Automatic Methods

This is the first release for **Automatic Methods**. **Automatic Methods** describe how to generate mesh for the Sheet and Sweepable bodies when no other methods are scoped to the bodies.

For more detailed information, see Automatic Methods.

3. Enhancements to Automatic (PrimeMesh)

Automatic (PrimeMesh) has been enhanced to:

- Provide **Partial Defeature** under **Advanced** to partially merge the geometry by suppressing only a portion of the edge joining two faces based on the **Thin Face Width**, **Sharp Angle** and **Feature Angle**.
- Provide **Repair Tolerance** under **Advanced** to repair and collapse the smallest edges of faces.

For more detailed information, see Automatic (PrimeMesh).

4. Enhancements to Mesh Workflows

Mesh Workflows has been enhanced to:

- Support Sizing
  - Provide **Create Size Field** step having the following sizing controls:
    - → **Constant Sizing**
    - → **Curvature Sizing**
    - → **Proximity Sizing**
    - → **Custom Names**
    - → **Body of Influence Sizing**
  - Provide **Size Field Wrapper** control under **Wrap** operation.
  - Provide **Size Field Surface Mesher** control under **Mesh Surface** operation.
  - Provide **Size Field Volume Mesher** control under **Mesh Volume** operation.
  - Provide **Size Fields Name** outcome.
- Create **HemiConvex Irregular Enclosure**.
- Provide Sizing wizard and icons for **Show Wizards**, **Selection Preview** in **Domain Browser**.

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 the geometry.

For more detailed information, see Patch Conforming Algorithm.

6. Enhancements to Mesh Quality Worksheet

Mesh Quality Worksheet has been enhanced to provide right-click options for **Quality Criterion**:

- **Visibility Off**

Mesh Quality Worksheet has been enhanced to provide icons to **Toggle the Contour** plotting between "classic" and "warning/error" types.

For more detailed information, see Mesh Quality Worksheet.

7. Enhancements to MultiZone Method

MultiZone Method has been enhanced to:

- Provide a new **Decomposition Type** named **Medial Axis** that identifies the medial axis path, constructs surface blocking for the sectional profile, and extrudes along the medial axis path or revolves about the axis to create a 3D hexa blocking.
- Provide a new **Surface Mesh Method** named **PrimeMesh Quad Dominant Method** that merges triangles into quads to create a surface mesh.

For more detailed information, see MultiZone Mesh Method.

8. Enhancements to Meshing Options

Meshing Options has been enhanced to include:

- **Default Sheet Body Method** to select the default method for the sheet bodies. Now, you can use **Automatic (PrimeMesh) Method** for automatic shell meshing and provide high performance.
- **Skip Part Remesh in Worksheet Meshing** to skip remeshing the failed rows in the worksheet in case of mesh failure and continues meshing the consecutive rows.

For more detailed information, see Meshing Options on the Options Dialog Box .

9. Enhancements to Repair Topology

Repair Topology is enhanced to provide **Partial Defeature** under **Advanced** to partially merge the geometry by suppressing only a portion of the edge joining two faces based on the **Thin Face Width**, **Sharp Angle** and **Feature Angle**. For more detailed information, see Repair Topology.

10. Enhancements to Sweep Method

Automatic Thin and **Manual Thin** sweep methods have been enhanced to support more than one division for multibody parts. For more detailed information, see Sweep Method Control.

ICEM CFD

Release 2025 R1 development efforts included enhancement of Ansys ICEM CFD as a standalone application as well as continued development of its underlying technology exposed within the Ansys Workbench-based Meshing application.

Ansys ICEM CFD 2025 R1 has no new features or enhancements.

Ansys Connect Products

Release notes are available for the following Ansys Connect products:

Ansys Minerva

The following topics provide an overview of what is new and improved in Ansys Minerva 2025 R1:

1. New Features and Enhancements

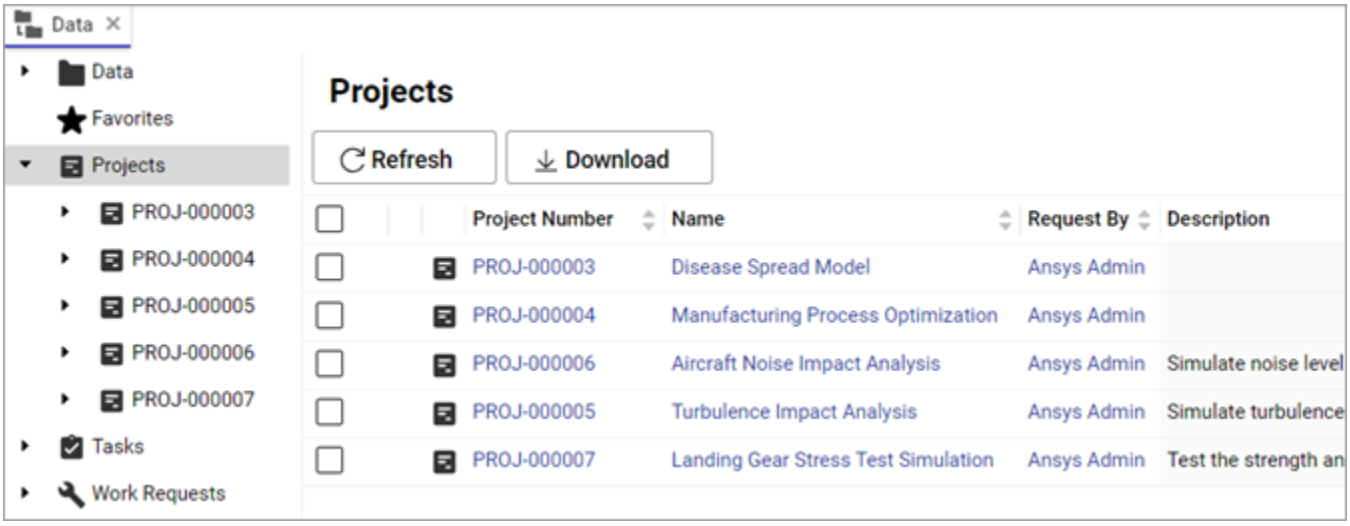
New features and enhancements are available in the following areas of Ansys Minerva 2025 R1:

1.1. Installation and Setup

This version of the Ansys Minerva solution is based on Aras Innovator Release 32. Some of the installation prerequisites and minimum system requirements have been updated. See Installation Prerequisites.

1.2. Data

- The ECAD viewer now includes the following features:
  - **Mini map**: enables you to see where you are in a design. This is particularly helpful when you have zoomed in to specific components of the design.
  - **Box zoom**: lets you quickly zoom in to desired area of the design by drawing a rectangular box.
- In **Data** view, when viewing project, work request, and task folders, you now see columns relevant to these items.



1.3. Applications

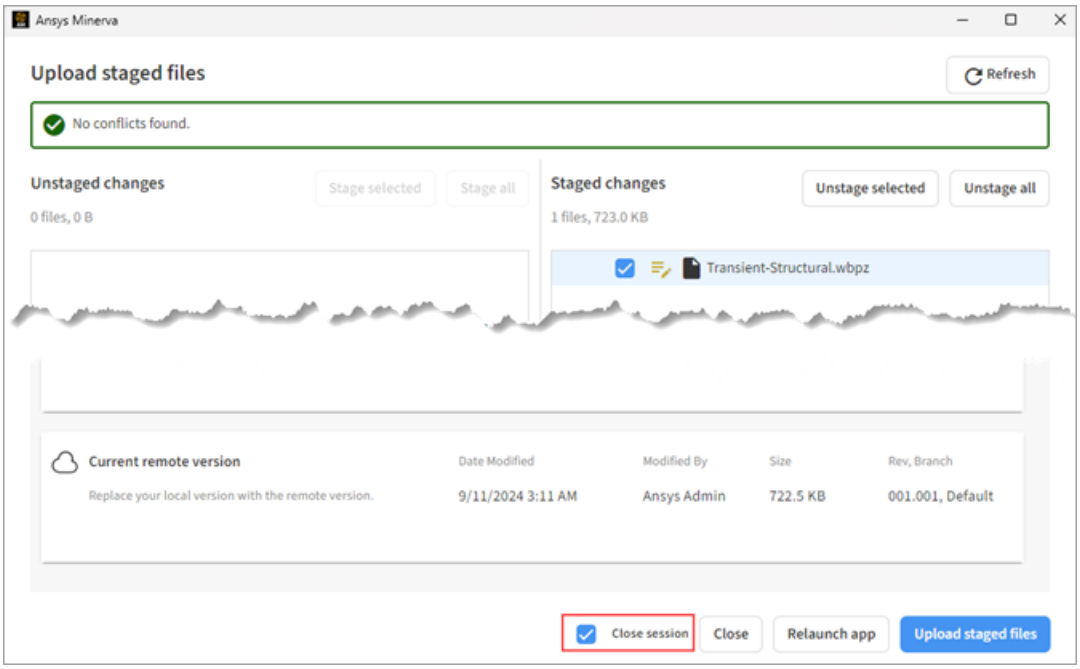
- You can now check and resolve conflicts in your files before uploading them back to the Ansys Minerva solution.

  Conflicts can occur when you download files locally and other users modify or delete the original files in the Ansys Minerva solution during collaborative work or when automated processes, such as scheduled jobs, update those files. Conflict management enables you to decide and preserve the correct version of files, whether it is your own or another user's, ensuring data integrity and efficient collaboration.

  For more information, see Resolving Conflicts in Files.

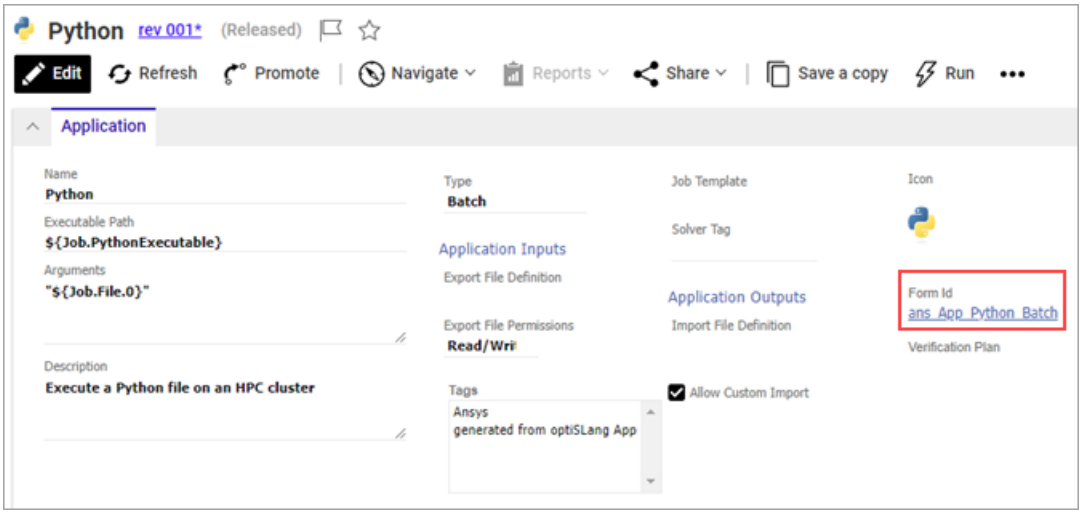
- You can now close your local application anytime and later resume your work in the same session by rerunning the application.

  To keep a session active, clear the **Close session** check box while uploading files. To resume your work, you can go to the related task and run the application again.



This is useful in events such as saving your work at the end of the day or releasing your application license so that a colleague can use it.

- Job templates are deprecated. Existing applications using job templates will continue to function as is. However, you should use **Forms 2.0** for designing forms for new applications. Some batch applications in the Ansys Minerva solution still use job templates. They will transition to Forms 2.0 in upcoming releases.



The Python batch application included with the Ansys Minerva solution now uses a new form created using Forms 2.0. You can use this form as a reference when creating forms for your new applications. See *Ansys Minerva Configuration Guide:* Creating Forms for Applications.

1.4. Configuration

- The Form Builder (Forms 2.0) has been enhanced in both functionality and styling, allowing you to create more advanced forms:

New components:

- Input Grid (`input-grid`): Arranges inputs in tabular format.
- Image (`img`): Lets you add representational images or icons on a form.
- Accordion (`viz-details`): Creates collapsible sections for better content organization.

Enhanced components:

- Form (`ans-form`) now allows you to specify width and height for forms displayed in dialogs, and a method to be called when any button on the form is clicked.
- Input: Search (`input-search`) now lets users select multiple items.
- Text Block (`div`) and Section (`section`) now supports styling customization using various properties.
- Column (`viz-column`) now lets you adjust the relative width of columns in the column set using width ratios.
- New properties, **Label Position** and **Label Width**, let you adjust the position of labels and control their width for relevant components.

- The `ElementType` component in document types has been extended to include the **Additional Data** (`additional_data`) property, which stores additional information about the element type in JSON format.

  You can use this property to provide configuration data specific to the element type. See *Ansys Minerva Configuration Guide:* Element Type (`ElementType`).

- Starting 2025 R1, the Ansys Icepak application, both as a standalone product and within the Ansys Workbench framework, is no longer available. Consequently, the Ansys Minerva solution no longer includes configurations related to the Ansys Icepak application.

  If you need to use an earlier version of the Ansys Icepak product with Ansys Minerva 2025 R1 or later, contact your support representative.

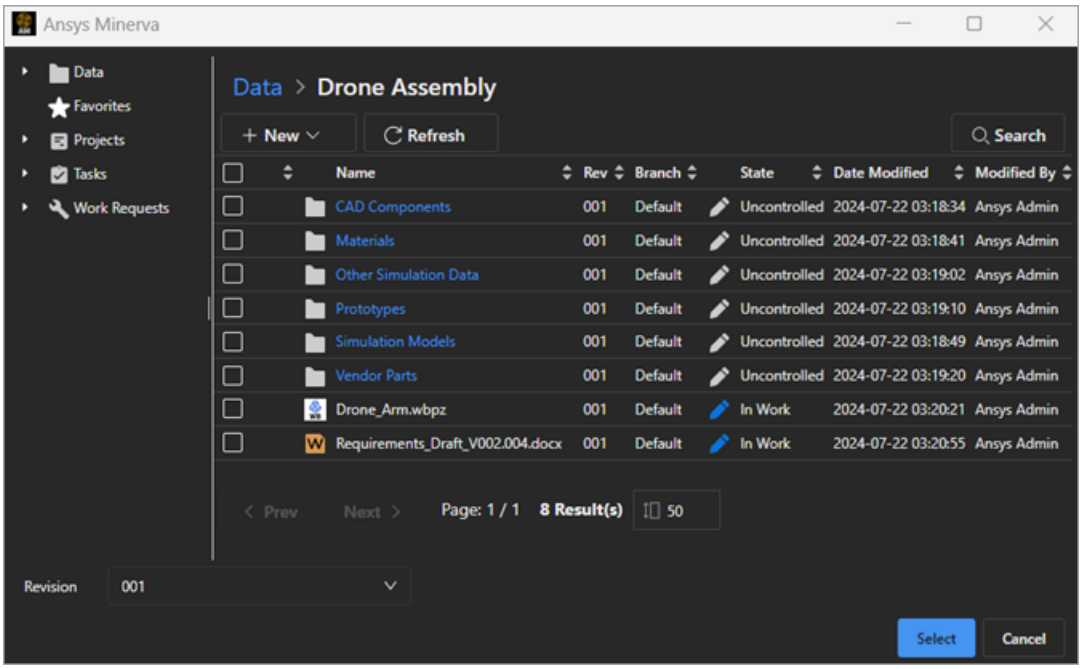
- Dependency detection now supports detecting dependent files located outside the subject file's folder. For example, if the dependent file is in a folder that is located alongside the subject file's folder, use `../folder_name/filename` to reference it in the parameter document.

1.5. Work Requests and Tasks

In the **Flow View** of a work request, you can now specify assignees for multi-user and milestone tasks using the **Properties** panel. See *Ansys Minerva User's Guide:* Assigning Tasks.

1.6. Generic Connector

- You can now use the Generic Connector with a Dark theme or a theme of your choice. Additionally, the user interface of the Generic Connector windows, such as file explorer and staging windows, has been enhanced for a better user experience.



To apply a theme, use the **`--ui-theme`** option with the Generic Connector commands. For more information, see *Ansys Minerva User's Guide:* Ansys Minerva Generic Connector Commands.

- You can now configure your working directory to exclude certain files from being considered in Generic Connector commands. See *Ansys Minerva User's Guide:* Ignoring Files.

  This enables you to manage your data more effectively by excluding files, such as temporary files, log files, local configuration files, files with sensitive information, or files that are irrelevant to your work.

- The **`upload`** and **`download`** commands now generate JSON output when they partially or completely fail. The output includes details of items that were successfully uploaded or downloaded, as well as items that failed.

  Schema:

```
{
  "items": [ /* Items that were uploaded or downloaded successfully */],
  "error": {
    ...
    "items": { /* Items that failed to download or upload */}
  }
}
```

  For more information, see *Ansys Minerva User's Guide:* JSON Output Schema.

- When using the **`download`** command, the new **`--filter`** option lets you filter items based on file extensions and certain item properties, such as classification, state, and revision. The specified filters appear in the file selection dialog box, making it easier to select the files you want to download.

  Additionally, the **`--filter`** option in the **`select-items`** command has also been enhanced to support item properties as filter criteria.

  For more information, see *Ansys Minerva User's Guide:* download and select-items.

- You can now generate help for Generic Connector commands in JSON format. For more information, see *Ansys Minerva User's Guide:* help.
- When uploading a folder using the **`--version-folders`** option, if a folder with the same name already exists in the **Uncontrolled** state at the destination, the uploaded version-controlled folder now replaces it.

2. Resolved Issues

For a list of issues that have been resolved in Ansys Minerva 2025 R1, see the Ansys Minerva section of the *Ansys Resolved Issues and Limitations*.

3. Known Issues and Limitations

All issues and limitations known at the time of release are listed in the Known Issues and Limitations section of the *Ansys Minerva Troubleshooting Guide*.

Issues discovered after release are listed in the Ansys Minerva section of the *Ansys Known Issues and Limitations*.

4. Documentation

The information related to *Job Templates*, *Legacy Forms*, and *Ansys optiSLang Web Service* has been moved to the *Deprecated Features* section of the *Configuration and User's guides*, as these features are now obsolete.

Distributed Compute Gateway (DCG)

The following topics provide an overview of what is new and improved in Distributed Compute Gateway 2025 R1:

1. New Features and Enhancements

No enhancements were made to Distributed Compute Gateway in this release.

2. Resolved Issues

For a list of issues that have been resolved in Distributed Compute Gateway 2025 R1, see the Distributed Compute Gateway (DCG) section of the *Ansys Resolved Issues and Limitations*.

3. Known Issues and Limitations

All issues and limitations known at the time of release are listed in the Troubleshooting section of the *Distributed Compute Gateway User's Guide*.

Issues discovered after release are listed in the Distributed Compute Gateway (DCG) section of the *Ansys Known Issues and Limitations*.

optiSLang

The following sections contain release information for optiSLang 2025 R1:

1. Web post-processing

Web post-processing has been updated:

- Can be freely integrated both with the customer environment and with Ansys products (for example, Ansys Minerva)
- Can now be used in Ansys Fluent (beta)
- Supports Dark and Light themes
- Interactive integration
  - Obtaining the currently selected design(s) and dimensions
  - Events that notify on design selection and dimension changes

2. App creation automation

New generation of autogenerated apps from Ansys optiSLang®.

- Capabilities included
  - Autogenerated web app with Pre/Post and Run capabilities
  - Provided in the oSL installer
  - Can be deployed and run in desktop, on-premises, or cloud-based platforms
  - User can modify the UI using python (Plotly Dash)
- Projects can be migrated from OWA into an autogenerated app:
  1. Unzip OWA

2. Open project.opf with newest Ansys optiSLang® version
3. Run "Generate App" wizard in oSL GUI or via command line option
4. Modify UI of autogenerated App if needed

- Low Code Customizations of AWA
- Added App generation via command line to automatically create AWA/OWA without a mouse
- Added Python API for Placeholder definition and modification
  - Project creation without GUI
  - Reduce error-prone tasks
  - Traceability of App creation

3. Algorithms

"Bring your own AI" - Signal Models (beta)

- Python API for building and serving external signal models in MOP
- Create and solve your own meta-modelling technique for signals
- Initial MOP competition for Signal models

Improved Performance -- Metamodels accelerated

- MOP Solver: Multidesign solve (default: 50)
- Available as setting in MOP Solver properties

FMU export of DFFN (beta)

- Supports Manual and Smart layouts
- FMU file loadable in the FMU node
- Approximation enabled

4. User Interface

New icons

- Better visibility in Light and Dark mode - Aligned with Ansys styling
- Added a configuration setting to use the legacy styling and icons

New project wizard

- Project is immediately saved to specified location. Avoids usage of temporary files
- OPR directly created. Easier use with reference files.
- Can also skip this step and save the project later.

Postprocessing Palette Editor

- moved from Beta
- Removed unneeded functionality
- Unified "New" and "Open" into one action button
- Added absolute values mode
- Added loading functionality
- Palettes can be loaded either from folder inside the installation or additional folder given per settings
- Added the ability to define a default palette in settings which will be applied when monitoring is opened
- Reworked actions to modify palette from action button to a button bar at the top of the palette tab

Excel Add-In

The Excel Add-in has been updated:

- MOPSolver functionality remains.
- A "Go to optiSLang" option was added to the ribbon.
- A "Go to optiSLang Postprocessing" option was added to the ribbon.
- OMDB creation was replaced with stronger functionality of postprocessing.

5. Connectors

The following connector enhancements were made:

- Ansys HPS Node node (beta)
  - Connect to Ansys HPS from an Ansys optiSLang® workflow.
- Ansys Fluent-Solver node
  - optiSLang inside Fluent -- Data Visualization using optiSLang's Postprocessing module (Beta)
- Remote Submission support

- Create SSH Remote profile for all combinations of Windows and Linux
- Use in Process and Script nodes

  → Input files will be copied

  → Execute process/script on remote machine

  → Output files will be copied back

Ansys Platform Products

Release notes are available for the following Ansys Platform products:

ACT

No enhancements were made to ACT in 2025 R1.

Ansys Dynamic Reporting

This document covers the Ansys Dynamic Reporting release notes for versions 2025 R1 and later. For previous release notes, see the Ansys Dynamic Reporting section in the relevant version of the EnSight release notes.

The following list highlights the new features and enhancements for Ansys Dynamic Reporting in 2025 R1.

- Added a new linear regression Statistical Analysis generator template to enable statistical analysis.
- Added a new Items Comparison generator template to enable data filtering and property settings.
- Added web components for reports with limited control over the report style.
- Added support for animations and 3D scenes in the Slider template.
- Added native support for histogram plots.
- Expanded the Data Filter layout template to support histograms and bar charts.
- Expanded PowerPoint export to support vertical, center and sub titles.
- Improvements in error messages for PowerPoint export and animation rendering.
- Expanded sort options to include natural sorting for the Slider layout template.
- Added a serverless configuration of Ansys Dynamic Reporting (beta release).

Ansys Viewer

The following sections contain release information for Ansys Viewer 2025 R1:

1. Advisory: Ansys Viewer End of Life

Ansys Viewer 2025 R1 will be the final version of Ansys Viewer released by Ansys. As of this release, no further updates or bug fixes will be provided.

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

If you use Ansys Viewer for graphics visualizations, Ansys encourages you to transition to alternative tools or workflows. If you have any questions or need assistance, please contact Ansys support.

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

2. New Features and Enhancements

No enhancements were made to Ansys Viewer in this release.

3. Resolved Issues and Limitations

See the Resolved Issues and Limitations document for information on previous limitations and key defects resolved in the current release.

4. Known Issues and Limitations

All issues and limitations known at the time of release are listed in the Known Issues and Limitations section of the *Ansys Viewer User's Guide*.

CAD Release Notes

This section summarizes the new features in CAD Integration Release 2025 R1.

For more information, see the CAD Integration section of the Ansys Help.

Rocky Linux 8.10 and 9.4 Support

The following interfaces are supported by Ansys:

- ACIS
- CATIA V5
- IGES
- Monte Carlo N-Particle
- NX (limited to version 8.10 and NX 2406)
- Parasolid
- STEP

Ubuntu LTS 20.04 and 22.04 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 2023 (Reader, Linux and Windows)
- AutoCAD 2019 (Reader, Windows only)
- CATIA V5 V5-6R2024 (Reader, Linux and Windows)

- CATIA V5 - (CADNexus CAPRI CAE Gateway-v4.10.0) V5-6R2024 (Reader, Windows only)
- Creo Parametric 11.0 (Reader, Windows only)
- IGES 5.3 (Reader, Linux and Windows)
- Inventor 2025 (Reader, Windows only)
- JT 10.9 (Reader, Windows only)
- NX 2406 (Reader, Windows only)
- NX 2406 (Plug-In, Linux and Windows)
- Parasolid 36.1 (Reader, Linux and Windows)
- Revit 2024 (Plug-In, Windows only)
- STEP AP242 (Reader, Linux and Windows)

For detailed version support information, see CAD Integration> Geometry Interface Support in the CAD Integration section of the Ansys Help.

Information about past, present and future CAD, operating system and platform support is viewable via the Ansys, Inc. website (Customer Care > Support> More Support > Platform Support).

Distributed Compute Services (DCS)

No enhancements were made to DCS in 2025 R1.

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 2025 R1:

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 Remote Solve Manager (RSM) section of the *Ansys Known Issues and Limitations*.

System Coupling

The following features have been added to the System Coupling 2025 R1 release:

1. Graphical User Interface

The following improvements have been made to System Coupling's graphical user interface (GUI):

- More options have been added to the menu bar and toolbar buttons.

2. Mapping Accuracy

The following improvements have been made to System Coupling's mapping capabilities:

- Numerical results for case involving the following mapping algorithms are expected to show small differences:
  - Profile-preserving mapping, volume-to-volume and volume-to-point-cloud (both nodal and elemental source)
  - Profile-preserving mapping, surface-to-surface and surface-to-point-cloud (both nodal and elemental source)
  - Any mapping with point cloud source

3. License Checking

System Coupling now performs a license check to ensure that licenses required to run a case are available. The check is performed the first time a command which starts the participants is called. If a license is not available, an error message is displayed.

4. Embedded Viewer

You can now view and interact with live visualization results in either the EnSight user interface, or the new embedded viewer in the System Coupling user interface.

5. 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*.

6. 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

The following general enhancements were made for 2025 R1:

- The **ACP-Post** system is now unsupported in Workbench. You can still run projects that include this system, however Ansys recommends updating the project to remove it.
- You can now transfer temperatures from a Steady-State Thermal or Transient Thermal system to an LS-DYNA system.
- SpaceClaim has been removed from the unified installer. You should plan to migrate your geometry preparation workflows to Ansys Discovery.
- You can now create Python based Workbench add-ins.
- The Ansys Motion PrepPost license option is now available in the **Model** cell properties **Licenses** list for the Motion system.

2. Engineering Data Workspace

For Release 2025 R1, the following enhancements have been made to the Engineering Data workspace:

- **Crack Growth - Paris' Law**. You can now directly choose the units of measurement for **Length per Cycle Units** and **Stress Intensity Factor Units** instead of using reference units. This allows you to enter experimental material data directly in your preferred unit system. Workbench and the solver automatically convert these values to the current unit system.
- The following material models for fatigue crack growth analyses are now available:
  - **Tabular Fatigue Law**
  - **Forman Equation**
  - **Walker Equation**
  - **NASGRO Equation V3**

– **NASGRO Equation V4**

3. Injection Molding Data

For Release 2025 R1, there are no new feature updates for Injection Molding Data.

Embedded Software Products

Release Notes

Release notes are available for the following Ansys embedded software products:

Scade One Release Notes

The following sections present the new features and enhancements for Scade One 2025 R1 release.

1. New Features and Enhancements

This section describes the new features and enhancements introduced in SCADE Suite 2025 R1 release.

Shared Web Licensing

- Web-based licensing, allowing easier deployments
- Better usage visibility (for customers and Ansys)

UI/UX Enhancements

- Align blocks
- Duplicate blocks
- Copy/paste assets
- Automatic generation of test harness
- New styling for group constructs

Enhancements and Bug Fixes

- Fixed limitations of code generator (incl. missing initialization analysis)
- Fixed limitations of debugger
- 50+ bugs fixed

SCADE Suite Release Notes

The following sections present the new features and enhancements for SCADE Suite 2025 R1 release.

1. New Features and Enhancements

This section describes the new features and enhancements introduced in SCADE Suite 2025 R1 release.

Code Generation for FMU Simulation

- Support of FMI/FMU 3.0 generation for “Model Exchange” and “Co-Simulation” modes including:
  - Support of arrays/structures
  - Extension of scalar data types

SCADE Suite Design Verifier

- Improved scenario generation including only input values relevant for property falsification

SCADE Suite UX Improvements

- Project reloading improvements allowing to reload all modified projects of the workspace in a single action

SCADE Automotive Package

- Ability to define multiple initialization runnables for a single software component
- Support for Diagnostic Service Dependency allowing to describe how diagnostic events are processed and stored:
  - Extended AUTOSAR configuration with Diagnostic Service Dependency concepts
  - Ability to import/export ARXML files, design models, and check model consistency with respect to new concepts
  - Wizard assistance for easy creation of several Diagnostic Service use cases
- Import of ARXML files from 4.0.1 to 4.8 versions

- New **ACG 2.7**:
  - Support for multiple initialization runnables for a single software component
  - Certification credits for ISO 26262 planned for next SCADE version
- Ability to set code generation using either **ACG 2.6** (with certification credits) or **ACG 2.7** (no certification credits)

SCADE Test Integration

For details, consult SCADE Test Release Notes (p. 253).

SCADE LifeCycle Integration

For details, consult SCADE LifeCycle Release Notes (p. 263).

2. Certification Credits

The SCADE Suite 2025 R1 release includes the following tool with certification credits:

- **SCADE Suite KCG 6.6.4** and **SCADE Suite KCG 6.6.2** for qualification under DO-178C/DO-330 up to TQL-1 and for certification under ISO 26262 up to ASIL D, IEC 61508 up to SIL 3, or EN 50128 up to SIL 3/4
- **SCADE ACG 2.6** for certification under ISO 26262 up to ASIL D
- See also Certification Credits (p. 254) for SCADE Test and Certification Credits (p. 263) for SCADE LifeCycle

3. Limitations and Workarounds

This section describes the limitations and workaround solutions applying to the SCADE Suite 2025 R1 release.

Limitation in Simulink Importer

- Import does not support tunable popup parameters set using a referencing enumeration class.

Limitation in Code Generation for FMUs

- When using FMU states, any modification of model requires to save FMU states.

Limitation in Python Scripts

- Some Python extension modules (Python modules implemented in C) like numpy or lxml are not working in SCADE environment.

Limitation in SCADE Suite Modeling

- Floating windows do not operate properly when a SCADE Architect project is loaded at the same time as SCADE Suite: close and reopen SCADE Suite to resolve the issue.

Limitation in SCADE Suite Simulation

- Ada language pragmas referenced in SCADE Suite KCG “pragma” pragma may potentially not be supported for Ada code simulation.

Limitations in Code Integration

- [Ada] Provided Targets and Adaptors do not support Ada. Ada dedicated utilities can be developed using Python API.
- [C] Imported types at root level are not supported when generating wrapping code for Ansys Twin Builder.

Limitation in Multicore Code Generator

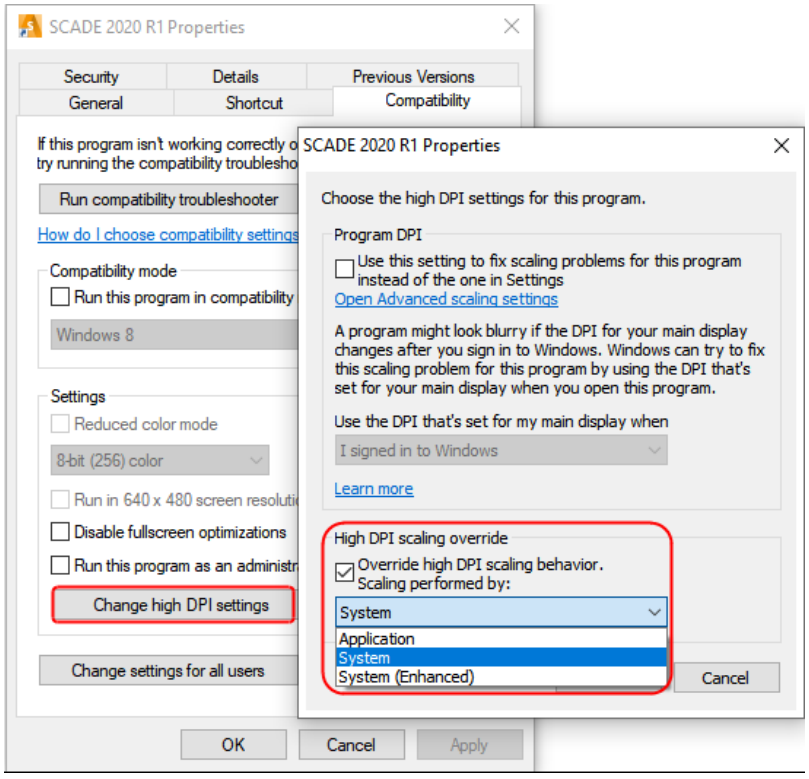
- With clocked tasks, generated code may contain cyclic dependency between methods, which prevents from integrating it.

Limitation in SCADE Applications

- SCADE applications may not scale well (with symptoms like tiny icons and UI elements) on high-DPI (high-dots-per-inch) displays, depending on the display scaling factor selected for the user's Windows display settings.

  Workaround*: users can let Windows automatically manage the scaling of SCADE applications. This may result in applications being blurry, but scaled correctly.

  1. Right-click the SCADE product binary (scade.exe) in File Explorer (or Desktop shortcut) to open Properties. On Compatibility tab, click **Change high DPI settings**, under High DPI scaling override section, check the **Override high DPI scaling behavior** option.
  2. Select **System** for the **Scaling performed by** drop-down.



* This is only applicable to Windows 10 version "1703" (also called "Creators Update", delivered in April 2017) and subsequent ones, and can be applied to all SCADE versions. The same operation can be applied to the vcs.exe binary in the same folder.

Limitation in PDF User Documentation

- Hyperlinks in PDF documents opened in some web browsers may not work, preventing navigation between documents.

  Workaround: set Adobe Acrobat as default PDF reader.

4. Migration Information

SCADE Suite 2024 R2 to 2025 R1

- No format change: no project migration issue

SCADE Suite 2024 R1 to 2024 R2

- No format change: no project migration issue
- Support for migrating Scade 6.4 projects to Scade 6.6 projects is no longer available. Loading Scade 6.4 models is no longer supported. Please contact your Ansys representative or Support if you have such need.
- In SCADE Suite Metrics and Rules Checker, the rule SCADE_R001 (for detecting non-coverable points) is replaced by a set of design and methodological rules. In case, you were using such a rule in your projects, it is recommended to launch Model Coverability analysis in SCADE Suite. If need be, the rule can be requested from support.

SCADE Automotive Package 2024 R2 to 2025 R1

- Migration to extended AUTOSAR configuration done automatically upon project loading - No specific issue: see SCADE Automotive Package (each release) (p. 246)

SCADE Automotive Package 2024 R1 to 2024 R2

- No format change: no project migration issue: see SCADE Automotive Package (each release) (p. 246)

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 <support@ansys.com>.

5. Supported External Tool Versions

This section describes the external tools supported by SCADE Suite 2025 R1 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 2025 R1 release.

1. Fixes and Enhancements

This release of SCADE Display is a maintenance release containing fixes and enhancements.

SCADE Display Python and Java API

- New API functions for table of fonts

SCADE Display Code Generation

- New SCADE Display KCG 6.7.2 with certification credits (see Certification Credits (p. 250))
  - Supported on Windows 10 and 11
  - New **-no_timestamp** command line option for generating files without date/time information
  - In command line, DGFX input files accepted in any order and files can be duplicated
  - Simplified exit codes
  - Code generation performance greatly improved for models with large numbers of NPlicators
- Code generation can use either **SCADE Display KCG 6.7.1** or **SCADE Display KCG 6.7.2**

Code Generation for FMU Simulation

- Support of FMI/FMU 3.0 generation for “Model Exchange” and “Co-Simulation” modes including:
  - Support of arrays
  - Extension of scalar data type

SCADE Test Integration

For details, consult SCADE Test Release Notes (p. 253).

SCADE LifeCycle Integration

- New Reporter for SCADE Display with certification credits

For details, consult SCADE LifeCycle Release Notes (p. 263).

2. Certification Credits

The SCADE Display 2025 R1 release includes the following tools with certification credits.

- **SCADE Display KCG Code Generator 6.7.2**: for qualification under DO-178C/DO-330 up to TQL-1 and for certification under ISO 26262, IEC 61508 up to SIL 3, or EN 50128 up to SIL 3/4
- **SCADE Display KCG Code Generator 6.7.1**: for qualification under DO-178C/DO-330 up to TQL-1 and for certification under ISO 26262, IEC 61508 up to SIL 3, or EN 50128 up to SIL 3/4
- See also Certification Credits (p. 254) for SCADE Test and Certification Credits (p. 263) for SCADE LifeCycle

3. Limitations and Workarounds

This section describes the limitations and workaround solutions applying to the SCADE Display 2025 R1 release.

Limitations in SCADE Display Modeling

- Effect containers are incompatible with glTF objects and should not be used with them
- No support of animations internal to glTF objects when using GltFImporter widget
- Masks on active areas are not supported

Limitations in Code Generation for FMUs

- When using FMU states, any modification of model requires to save new FMU states
- When using FMU states, user imported function with memory must not store pointers

Limitation in SCADE Display Simulation

- “Restart” command and “Reset” scenario command in reference object simulation: array and structure variables are not updated to the Variable Dictionary's initial value. Instead, they are initialized to 0

Limitations in SCADE Display KCG & OGLX

- When drawing a primitive with arc segments, and setting transparency on the outline part of the primitive, darker points appear where the elementary lines used to draw the segment are superposed.
- Clip-planes are not supported in OGLX with OpenGL driver SC 1.0.1.
- Only 16 nested containers with Push/Pop matrix supported with OpenGL driver SC 1.0.1.

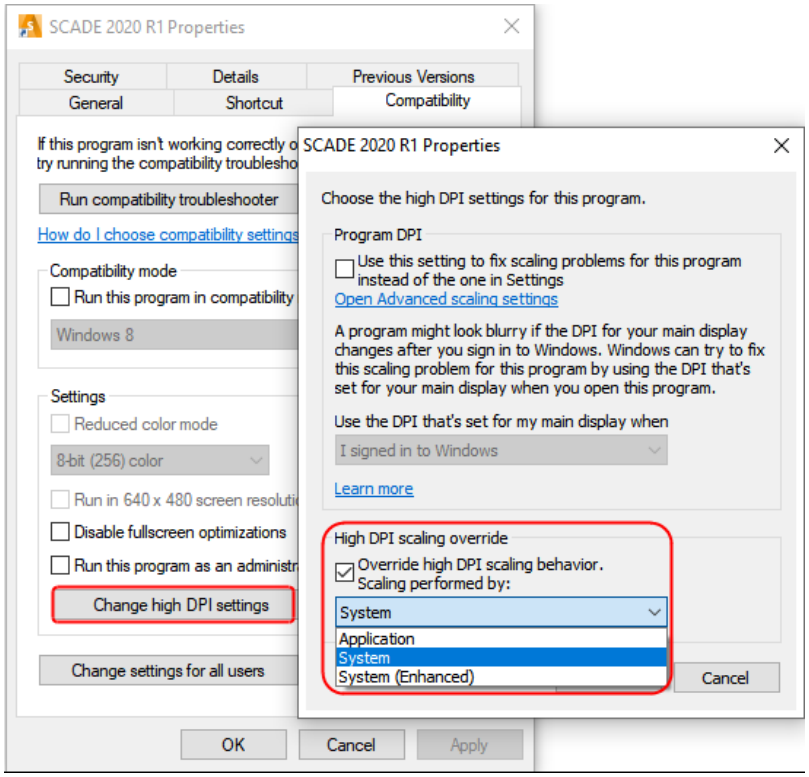
Limitation in SCADE Applications

- SCADE applications may not scale well (with symptoms like tiny icons and UI elements) on high-DPI (high-dots-per-inch) displays, depending on the display scaling factor selected for the user's Windows display settings.

  Workaround*: users can let Windows automatically manage the scaling of SCADE applications. This may result in applications being blurry, but scaled correctly.

  1. Right-click the SCADE product binary (scade.exe) in File Explorer (or Desktop shortcut) to open Properties. On Compatibility tab, click **Change high DPI settings**, under High DPI scaling override section, check the **Override high DPI scaling behavior** option.

  2. Select **System** for the **Scaling performed by** drop-down.



  * This is only applicable to Windows 10 version "1703" (also called "Creators Update", delivered in April 2017) and subsequent ones, and can be applied to all SCADE versions. The same operation can be applied to the vcs.exe binary in the same folder.

Limitation in PDF User Documentation

- Hyperlinks in PDF documents opened in some web browsers may not work, preventing navigation between documents.

  Workaround: set Adobe Acrobat as default PDF reader.

4. Migration Information

SCADE Display 2024 R2 to 2025 R1

- No format change: no project migration issue

- For projects created before 2025 R1 and loaded in 2025 R1, whenever editing existing Code Generation configuration settings, the configuration information in ETP file is updated as KCG671 for @SDY:CODEGENERATOR property

SCADE Display 2024 R1 to 2024 R2

- No format change: no project migration issue

SCADE Display 2023 R2 to 2024 R1

- Format change: at loading, automatic migration of `.etp` project files

For migration information related to previous versions, contact <support@ansys.com>.

5. Supported Graphical Libraries Version

This section describes the graphical libraries supported by SCADE Display 2025 R1 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 2025 R1 release.

1. New Features and Enhancements

This section describes the new features and enhancements introduced in SCADE Test 2025 R1 release.

Model Testing on Host for SCADE Display

- Improved test reports with display of all image files including filtered images used during image check

Model Coverage for SCADE Display

- Model coverage available with SCADE Display KCG 6.7.1 and SCADE Display KCG 6.7.2

Model Coverage for SCADE Suite

- Easier coverage resolution relying on dedicated script allowing to propagate justification of equivalent points

Model Coverability Analysis

- New complete test case generation capability including expected results (oracles) generation
- New optional capability for the generation of scenarios including only relevant inputs to reach coverage points
- New optional capability allowing to include justified points in coverability analysis.
- Improved information display in Model Coverage summary reports and in Model Coverability Analysis reports to help distinguish “Not Coverable” points as detected during Model Coverage or Coverability Analysis sessions

SCADE Test Tools for SCADE Display

- Renewed certification credits for SCADE Test Model Coverage for SCADE Display
- Renewed certification credits for SCADE Test Environment for Host for SCADE Display

- Renewed certification credits for SCADE Test Target Execution for SCADE Display
- For details, see Certification Credits (p. 254)

SCADE Rapid Prototyper

- Support of FMI/FMU 3.0 generation for “Model Exchange” and “Co-Simulation” modes including
  - Support of arrays
  - Extension of scalar data type

SCADE LifeCycle Integration

For details, consult SCADE LifeCycle Release Notes (p. 263).

2. Certification Credits

The SCADE Test 2025 R1 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 - 2025 R1** for qualification under DO-178C/DO-330 TQL-5 and for certification under ISO 26262 TCL3, IEC 61508 T2, or EN 50128 T2
- **SCADE Test Model Coverage for SCADE Display - 2025 R1** (for **SCADE Display KCG 6.7.2** usage) for qualification under DO-178C/DO-330 TQL-5 and for certification under ISO 26262 TCL3, IEC 61508 T2, or EN 50128 T2 (Ada coverage is outside certification scope)
- **SCADE Test Model Coverage for SCADE Display - 2021 R2**(for **SCADE Display KCG 6.7.1** usage) for qualification under DO-178C/DO-330 TQL-5 and for certification under ISO 26262 TCL3, IEC 61508 T2, or EN 50128 T2 (Ada coverage is outside certification scope)
- **SCADE Test Target Execution for SCADE Display - 2025 R1** for qualification under DO-178C/DO-330 TQL-5 and for certification under ISO 26262 TCL3, IEC 61508 T2, or EN 50128 T2

3. Limitations and Workarounds

This section describes the limitations and workaround solutions applying to the SCADE Test 2025 R1 release.

Limitation in Coverability Analysis

- Internal failure occurs when analyzing a model containing an operator abstracted by mixed list of imported or non-imported operators and where several abstract operators are polymorphic in type and size.

  Workaround:

  1. Update the model so that an operator abstracts either imported operators or non-imported operators.
  2. Update the model so that it does not contain any size polymorphic abstract operator.

Limitations in Rapid Prototyper

- No support of animations internal to glTF objects when using GltFImporter widget
- Masks on active areas are not supported
- No support for SCADE Suite KCG 6.6 Ada
- Code Generation for FMUs:
  - When using FMU states, any modification of model requires to save FMU states
  - When using FMU states, user imported function with memory must not store pointers

Limitation in Test Execution

- Snapshots not supported by test execution on SCADE Display specifications

Limitations in Test Target Execution

- **Test Harness Generation** (C or Ada): assignments and tests on data of external types, and snapshots are not supported
- **Test Harness for RTRT target** (C): for generated test harnesses to be properly interpreted by RTRT, `kcg_char` must be mapped to `unsigned char`
- **Test Harness for VectorCAST target**:
  - Models with sensors redefinition cannot be used by VectorCAST test harnesses (C code)
  - For generated test harnesses in C code to be properly interpreted by VectorCAST, `kcg_char` must be mapped to char and `kcg_true` must be mapped to 1 and kcg_false to 0 (default mapping)

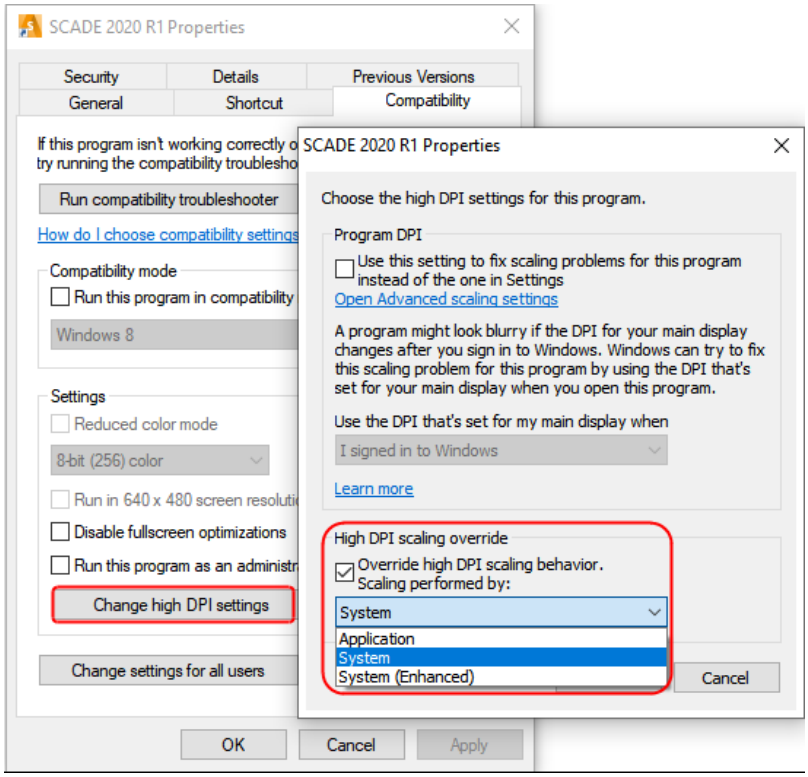
Limitation in SCADE Applications

- SCADE applications may not scale well (with symptoms like tiny icons and UI elements) on high-DPI (high-dots-per-inch) displays, depending on the display scaling factor selected for the user's Windows display settings.

  Workaround*: users can let Windows automatically manage the scaling of SCADE applications. This may result in applications being blurry, but scaled correctly.

1. Right-click the SCADE product binary (scade.exe) in File Explorer (or Desktop shortcut) to open Properties. On Compatibility tab, click **Change high DPI settings**, under High DPI scaling override section, check the **Override high DPI scaling behavior** option.

2. Select **System** for the **Scaling performed by** drop-down.



* This is only applicable to Windows 10 version "1703" (also called "Creators Update", delivered in April 2017) and subsequent ones, and can be applied to all SCADE versions. The same operation can be applied to the vcs.exe binary in the same folder.

Limitation in PDF User Documentation

- Hyperlinks in PDF documents opened in some web browsers may not work, preventing navigation between documents.

  Workaround: set Adobe Acrobat as default PDF reader.

4. Migration Information

SCADE Rapid Prototyper 2024 R2 to 2025 R1

- No format change: no project migration issue

SCADE Rapid Prototyper 2024 R1 to 2024 R2

- No format change: no project migration issue

SCADE Rapid Prototyper 2023 R2 to 2024 R1

- Format change: at loading, automatic migration of `.etp` project files

SCADE Test 2024 R2 to 2025 R1

- No format change: no project migration issue

SCADE Test 2024 R1 to 2024 R2

- No format change: no project migration issue

SCADE Test 2023 R2 to 2024 R1

- No format change: no project migration issue

For migration information related to previous versions, contact <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	2025 R1 (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) 6.7.2 (certification credits)
SCADE Test Model Coverage	2025 R1 (Display) 2024 R1 (Suite)
SCADE Test Target Execution	2025 R1 (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 2025 R1 release.

1. Fixes and Enhancements

This release of SCADE Architect is a maintenance release containing fixes.

SCADE Architect Modeling

- Cameo and Enterprise Architect model import:
  - Support for Sparx Systems EA 16.1
  - Support for Dassault Systèmes® Cameo Systems Modeler™ 2024x

SCADE LifeCycle Integration

For details, consult SCADE LifeCycle Release Notes (p. 263).

2. Limitations and Workarounds

This section describes the limitations and workaround solutions applying to the SCADE Architect 2025 R1 release.

Limitations in SCADE Architect Modeling

- Changing type of part/reference not supported
- Turning part into reference not supported
- SCADE Architect diagrams may not display connections and associations properly (for example, alignment of wires shifting) when opening diagram with different DPI (high-dots-per-inch) displays: the option **DPI Rendering** can be used in diagram properties to correct alignment
- Definition of new Guard and Weight properties on existing Control Flows may not display in Activity diagrams: recreate the Control Flow to refresh diagram with new property information.

Limitations in SCADE Architect Application

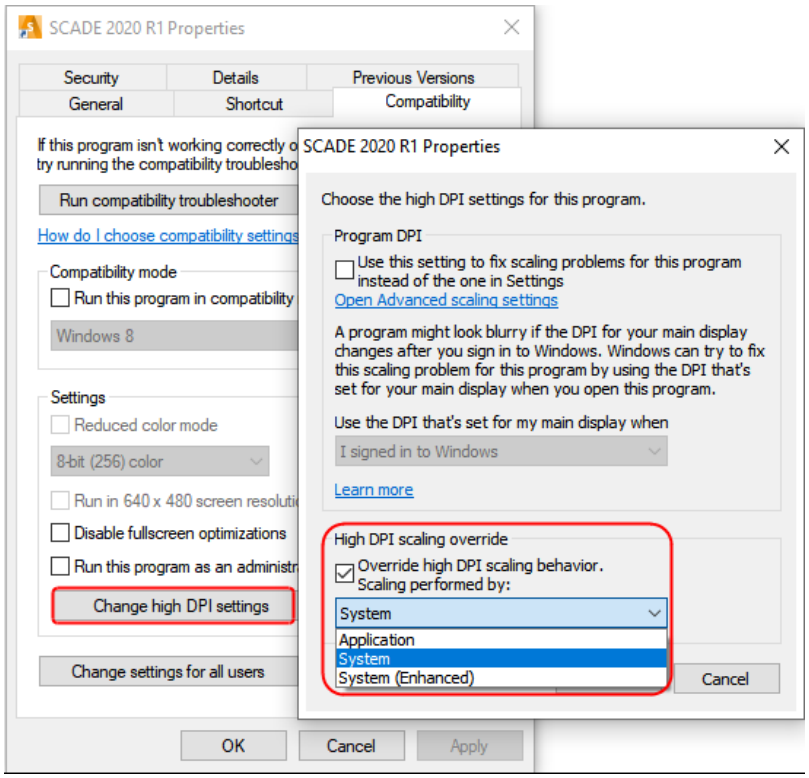
- SCADE Architect may fail to start or may allocate too little memory to its Java Virtual Machine under some rare circumstances (mostly when running in a virtualized environment with machines having less than 8 GB RAM). Default memory allocation can be fine-tuned by adding a specially crafted `sc-sys.ini` file in the `<INSTALLATION>/SCADE/` folder and containing the following line: `-Xmx1024m`. The value after `-Xmx` is given in megabytes and can be adjusted as required.

- SCADE applications may not scale well (with symptoms like tiny icons and UI elements) on high-DPI (high-dots-per-inch) displays, depending on the display scaling factor selected for the user's Windows display settings.

  Workaround*: users can let Windows automatically manage the scaling of SCADE applications. This may result in applications being blurry, but scaled correctly.

  1. Right-click the SCADE product binary (`scade.exe`) in File Explorer (or Desktop shortcut) to open Properties. On Compatibility tab, click **Change high DPI settings**, under High DPI scaling override section, check the **Override high DPI scaling behavior** option.

  2. Select **System** for the **Scaling performed by** drop-down.



  * This is only applicable to Windows 10 version "1703" (also called "Creators Update", delivered in April 2017) and subsequent ones, and can be applied to all SCADE versions. The same operation can be applied to the vcs.exe binary in the same folder.

Limitations in Model Import/Export

- Table query scripts, ATY files, and content providers must be explicitly added by users when exporting an archive

- Object colors in diagrams are lost when importing Enterprise Architect models

Limitations in Configurator

- Reference establishment is limited to the scope of user metaclasses. It is not possible to directly reference Architect API metaclasses.
- SCADE LifeCycle Reporter ignores configuration aspects and reports configured models as standard SCADE Architect models.
- During a working session, it is required to open only SCADE Architect projects or only SCADE Architect Configurator projects. If ever alternating projects, one must close the current session and reopen the application. A message warns you about potential conflicts or loading errors whenever switching between SCADE Architect projects and SCADE Architect Configurator projects in the same session.

Limitations in Diff/Merge

- Diagrams, Sequences, and Use Cases are only matched/ merged as a whole (no matching/merging of sub-elements).
- Notes with structured types are accessible as a whole from API and merged as a whole.

Limitation in Architect API

- SCADE Architect Java API for annotations depends on ANTLR 3.2.0 (Java component parsing textual files). The API cannot be installed on an Eclipse target platform that includes a more recent and incompatible version of the **org.antlr.runtime** plug-in. Recent versions of the XText Eclipse modeling component rely on ANTLR 4 and cannot be installed on the same platform as the API.
- SCADE Architect API has no support for concepts coming from Use Case or Sequence diagrams.

Limitation in PDF User Documentation

- Hyperlinks in PDF documents opened in some web browsers may not work, preventing navigation between documents.

  Workaround: set Adobe Acrobat as default PDF reader.

3. Migration Information

SCADE Architect 2024 R2 to 2025 R1

- No specific issue: see SCADE Architect (each release) (p. 261)

SCADE Architect 2024 R1 to 2024 R2

- No specific issue: see SCADE Architect (each release) (p. 261)

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 <support@ansys.com>.

4. Supported External Tool Versions

This section describes the external tools supported by SCADE Architect 2025 R1 release.

External Tools	Version
IBM® Rational® Rhapsody®	8.2
Sparx Systems Enterprise Architect	Up to 16.1
Dassault Systèmes® Cameo Systems Modeler™	Up to 2024x
Eclipse (for Architect API)	Oxygen 3a

SCADE LifeCycle Release Notes

The following sections present the new features and enhancements for SCADE LifeCycle 2025 R1 release.

1. New Features and Enhancements

This section describes the new features and enhancements introduced in SCADE LifeCycle 2025 R1 release.

SCADE LifeCycle ALM Gateway

- Ability to filter the traceability information to import from PTC Codebeamer™ requirement documents
- Ability to select requirements baselines with Siemens® Polarion® connection
- New Python script example for model export customization

SCADE LifeCycle Reporter

- New Reporter for SCADE Display with certification credits (see Certification Credits (p. 263))

2. Certification Credits

The SCADE LifeCycle 2025 R1 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 2025 R1 for SCADE Display Certification Data**
  - For qualification under DO-178C (DO-330 TQL-5)
  - For certification under ISO 26262, IEC 61508, and EN 50128
- **SCADE LifeCycle Reporter 2020 R2c for SCADE Test Certification Data**

  - For qualification under DO-178C (DO-330 TQL-5)
  - For certification under ISO 26262, IEC 61508, and EN 50128

- **SCADE LifeCycle Reporter 2024 R1 for SCADE ARINC 661 UA DF Generator Certification Data**
  - For qualification under DO-178C (DO-330 TQL-5)

- **SCADE LifeCycle Model Change 2021 R2 for SCADE Suite Certification Data Rev B**
  - For qualification under DO-178C (DO-330 TQL-5)
  - For certification under ISO 26262, IEC 61508, and EN 50128

3. Limitations and Workarounds

This section describes the limitations and workaround solutions applying to the SCADE LifeCycle 2025 R1 release.

Limitations in Requirements Management

- Exporting models to DOORS NG, Jama, or Polarion is slow (API-related issue from each ALM tool).

  Workaround: use custom model export from UI to reduce elements to export and launch export in batch.

- **Codebeamer Connections**:
  - If API throttling is active in Codebeamer ALM tool, import/export is slower.

- **Polarion Connections**:
  - When importing a Polarion document in a baseline, if the document contains both work items and headings at the same level, they might be ordered differently than in Polarion because of equal outline numbers.
  - Filtered documents cannot be imported if work items are referenced from another document (Polarion API issue)
  - Impossible to delete exported elements: Polarion API adds a “TO_DELETE” tag on such elements.

    Workaround: filter elements tagged by "TO_DELETE".

- **DOORS NG Connections**:
  - Selecting views is not available in DOORS Next Generation 6.0.5 & 6.0.6 connection, if Jazz Team Server has no Global Configuration Manager installed.
  - When working with DOORS NG 6.0.4, accessing the local configuration of DOORS NG is limited to the content of the initial stream.

    Workaround: fixed by using 6.0.5 or higher.

- **DOORS 9.x (native) Connections**:

  - When using DOORS baselines for traceability, requirement location by navigation from SCADE projects to ALM tools is not possible.

- **SCADE Suite**:

  - It is not possible to trace model objects defined textually in `.scade` files. [CR110-44]

  - Exporting model objects defined textually in `.scade` files causes object duplication or errors in DOORS, DOORS NG, Polarion, or Jama tools.

- **SCADE Architect**:

  - When using default full export, it is not possible to export back to ALM tool databases an AUTOSAR, Avionics, FACE, or any SCADE Architect configured model. Instead use custom model export.

Limitation in Reporter

- **SCADE Architect**: configured models are reported as standard SCADE Architect models and configuration aspects are ignored.

- **SCADE Suite**, **SCADE Architect**, and **SCADE Test**: Use of Unicode UTF-8 option in Windows Regional settings is not supported for report generation.

Limitation in PDF User Documentation

- Hyperlinks in PDF documents opened in some web browsers may not work, preventing navigation between documents.

  Workaround: set Adobe Acrobat as default PDF reader.

4. Migration Information

ALM Gateway 2024 R2 to 2025 R1

- No format change: no project migration issue

ALM Gateway 2024 R1 to 2024 R2

- When exporting models with images, all images (not exported previously due to a bug) are exported at once for the first export using the new release.

ALM Gateway 2023 R2 to 2024 R1

- No format change: no project migration issue

For migration information related to previous versions, contact <support@ansys.com>.

5. Supported External Tool Versions

This section describes the external tools supported by SCADE LifeCycle 2025 R1 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 9.16.0
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 2025 R1 Solutions for ARINC 661 release.

1. New Features and Enhancements

This section describes the new features and enhancements introduced in SCADE 2025 R1 Solutions for ARINC 661 release.

SCADE ARINC 661 Widget Library

- Support of CursorEventsExtension for all applicable widgets.

SCADE ARINC 661 Server

- Tap gesture optimization: now the tap event is taken into account immediately for widgets that do not use double-tap
- New server configuration flag to control each layer activation by the server

SCADE LifeCycle Integration

- For details, consult SCADE LifeCycle Release Notes (p. 263).

2. Certification Credits

The SCADE 2025 R1 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. 263) for SCADE LifeCycle Reporter for SCADE ARINC 661 UA DF Generator

3. Limitations and Workarounds

This section describes the limitations and workaround solutions applying to the SCADE 2025 R1 Solutions for ARINC 661 release.

Limitation in A661 Server Compiled with ANGLE

- On a laptop computer with Intel integrated graphics (driver version 27.20.100.8336 or upper), the server may freeze after few minutes if it is run on this graphics chip. Please contact Ansys SCADE Support in case this issue occurs.

Limitation in SCADE UA Page Creator

- The following widgets have no preview for display in Look Definition Preview: ActiveArea, CursorOver, MapHorz_ItemList, MapVert_ItemList, MapHorz_Container, SizeToFitContainer, ShuffleToFitContainer, TabbedPanel

Limitations in 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.

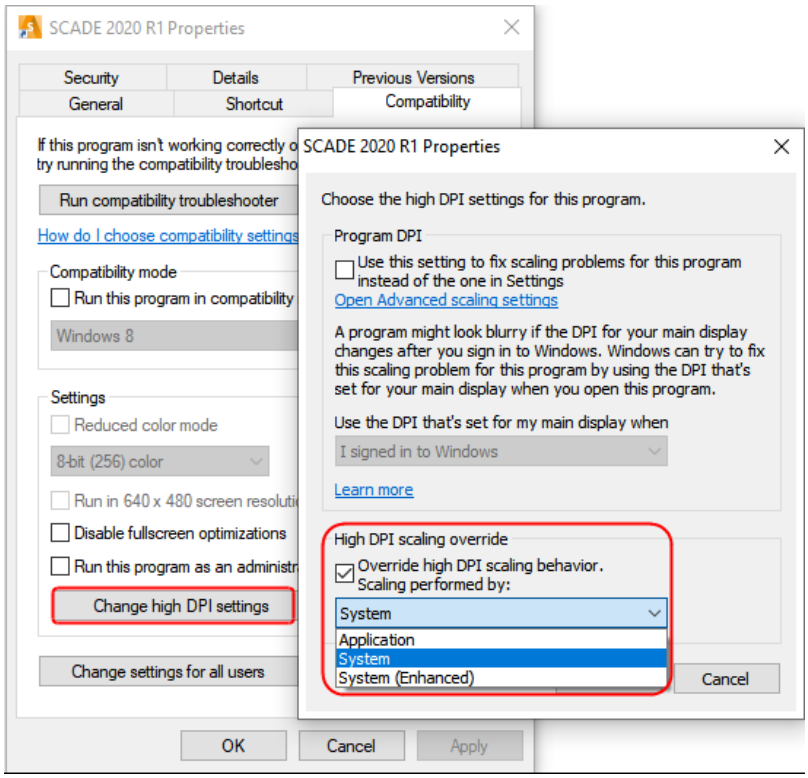
Limitation in SCADE Applications

- 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.

Limitation in PDF User Documentation

- Hyperlinks in PDF documents opened in some web browsers may not work, preventing navigation between documents.

  Workaround: set Adobe Acrobat as default PDF reader.

4. Migration Information

SCADE 2024 R2 to 2025 R1 Solutions for ARINC 661

- No format change: no project migration issue

SCADE 2024 R1 to 2024 R2 Solutions for ARINC 661

- UAPC models using the default A661 Configuration files: automatic migration from A661-6 to A661-9 library format upon loading. For some widgets having modified parameters, their value is lost and the default value is associated with the new parameter:

  – CheckButton: parameter PicturePosition is renamed as CheckboxPosition and is set to A661_LEFT

  – SymbolAnimated: LoopType renamed as LoopFlag and is set to A661_LOOP_FORWARD

  For Constants: various categories renamed, constant type is set to uncategorized

SCADE 2023 R2 to 2024 R1 Solutions for ARINC 661

- Format change: at loading, automatic migration of UAPC projects
- Server Creator projects: automatic split of widget capacities/definitions into separate files

For migration information related to previous versions, contact <support@ansys.com>.

SCADE Avionics Package Release Notes

The following sections present the new features and enhancements for SCADE Avionics Package 2025 R1 release.

1. Fixes and Enhancements

This release of SCADE Avionics Package is a maintenance release containing fixes and enhancements..

FACE Modeling

- Support for new SDM releases:
  - SDM v3.1.10 for FACE 3.1
  - SDM v3.0.10 for FACE 3.0
  - SDM v2.1.46 for FACE 2.1

2. Limitations and Workarounds

This section describes the limitations and workaround solutions applying to the SCADE Avionics Package 2025 R1 release.

Limitations in SCADE Avionics Package

- Due to CTS known issue, the char "0" should not be used in package naming. CTS checks fail and complain about missing .h/.hpp files if ever using this character in package naming.

  Workaround: use CHK_394 to detect and fix invalid naming of concerned elements.

- **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.

Limitation in PDF User Documentation

- Hyperlinks in PDF documents opened in some web browsers may not work, preventing navigation between documents.

  Workaround: set Adobe Acrobat as default PDF reader.

3. Migration Information

SCADE Avionics Package 2024 R2 to 2025 R1

- No specific issue: see SCADE Avionics Package (each release) (p. 272)

SCADE Avionics Package 2024 R1 to 2024 R2

- No specific issue: see SCADE Avionics Package (each release) (p. 272)

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 <support@ansys.com>.