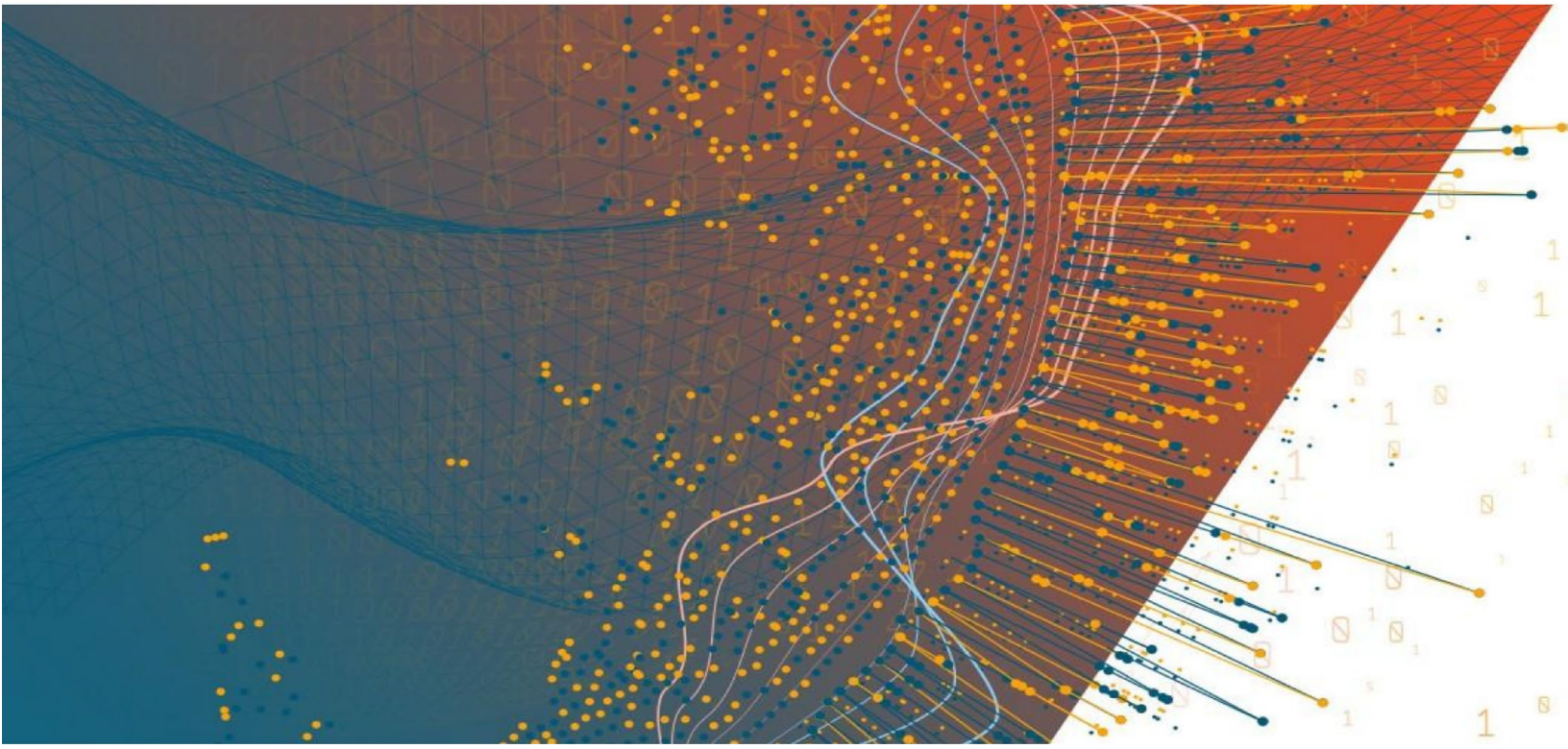


Altair® Inspire™ 2025.1

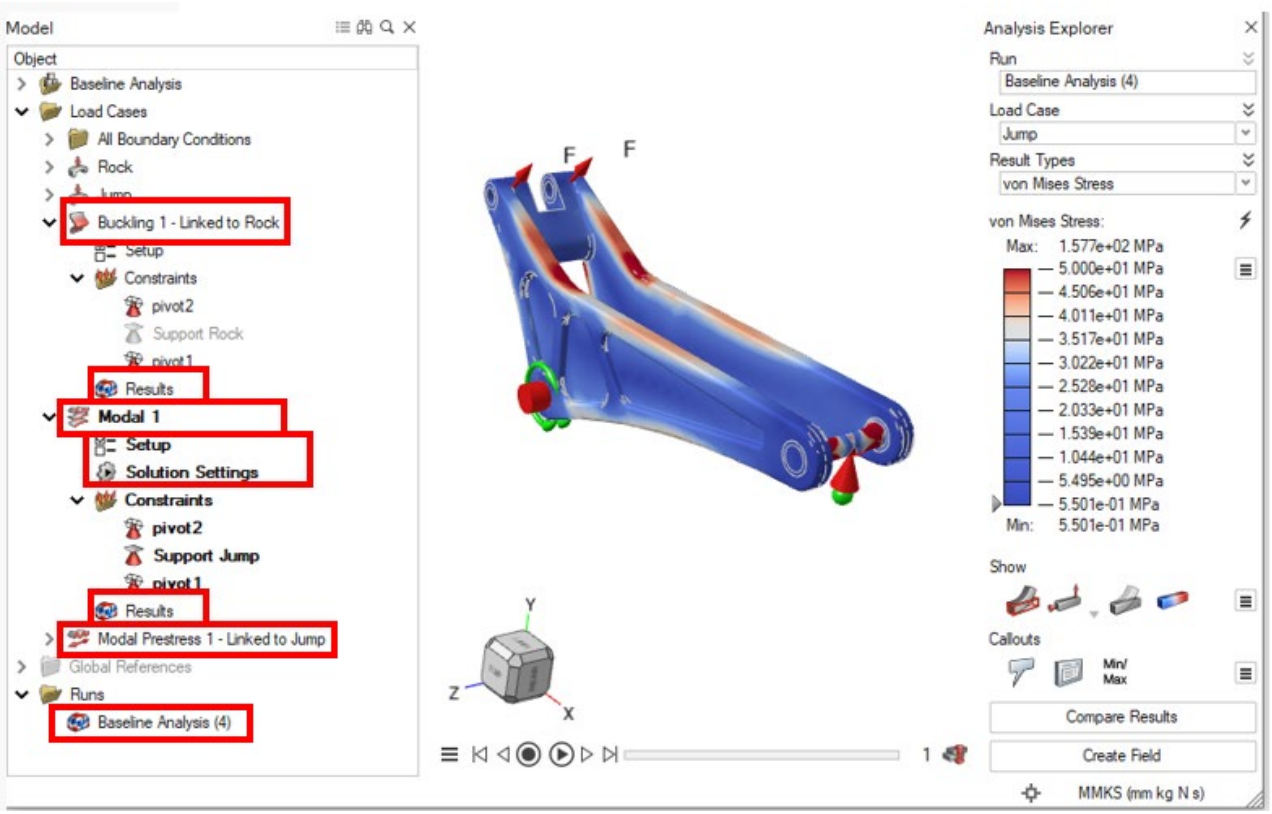
RELEASE NOTES

NEW FEATURES

Structure

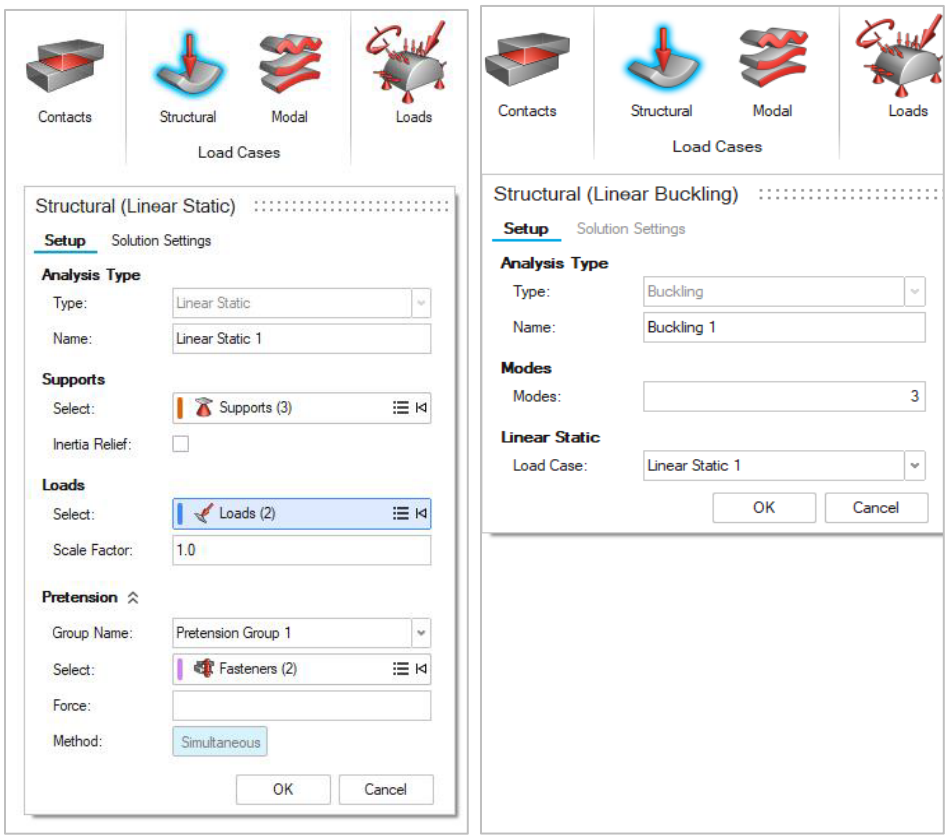
Run Management

Improved creation and editing of multiple analysis types with results in the Model Browser.

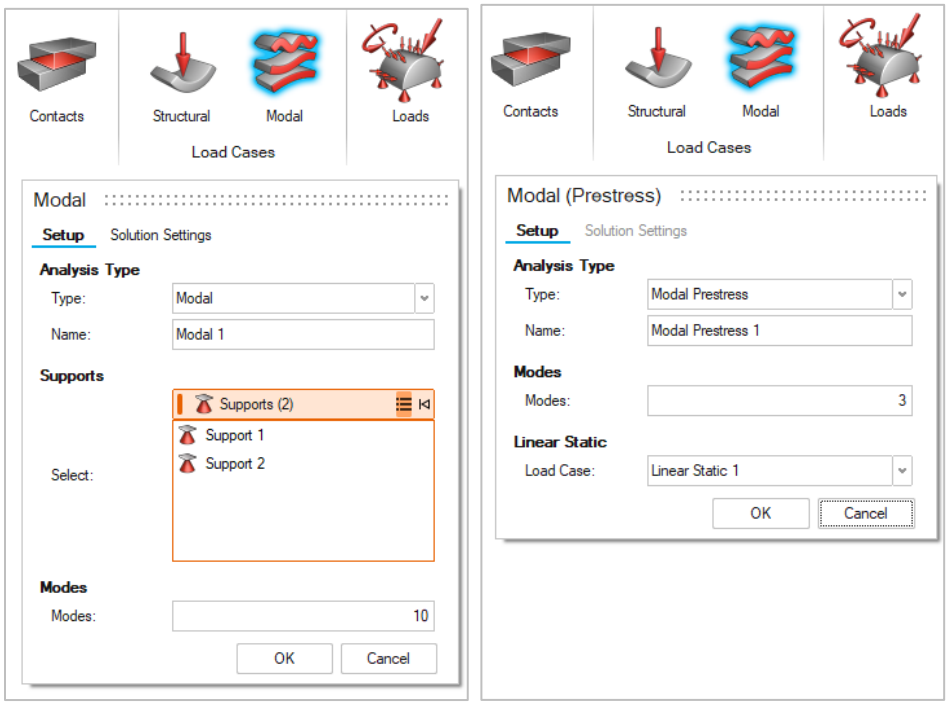


For more information, see Manage Runs.

With the new **Structural Load Cases**, you can create and edit linear static and buckling loads.



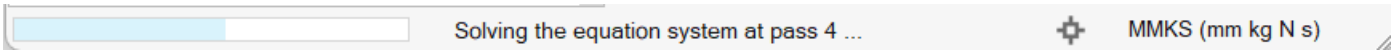
With the **Modal Load Cases** tool, you can create and edit modal and modal prestress loads.



For more information, see Load Cases.

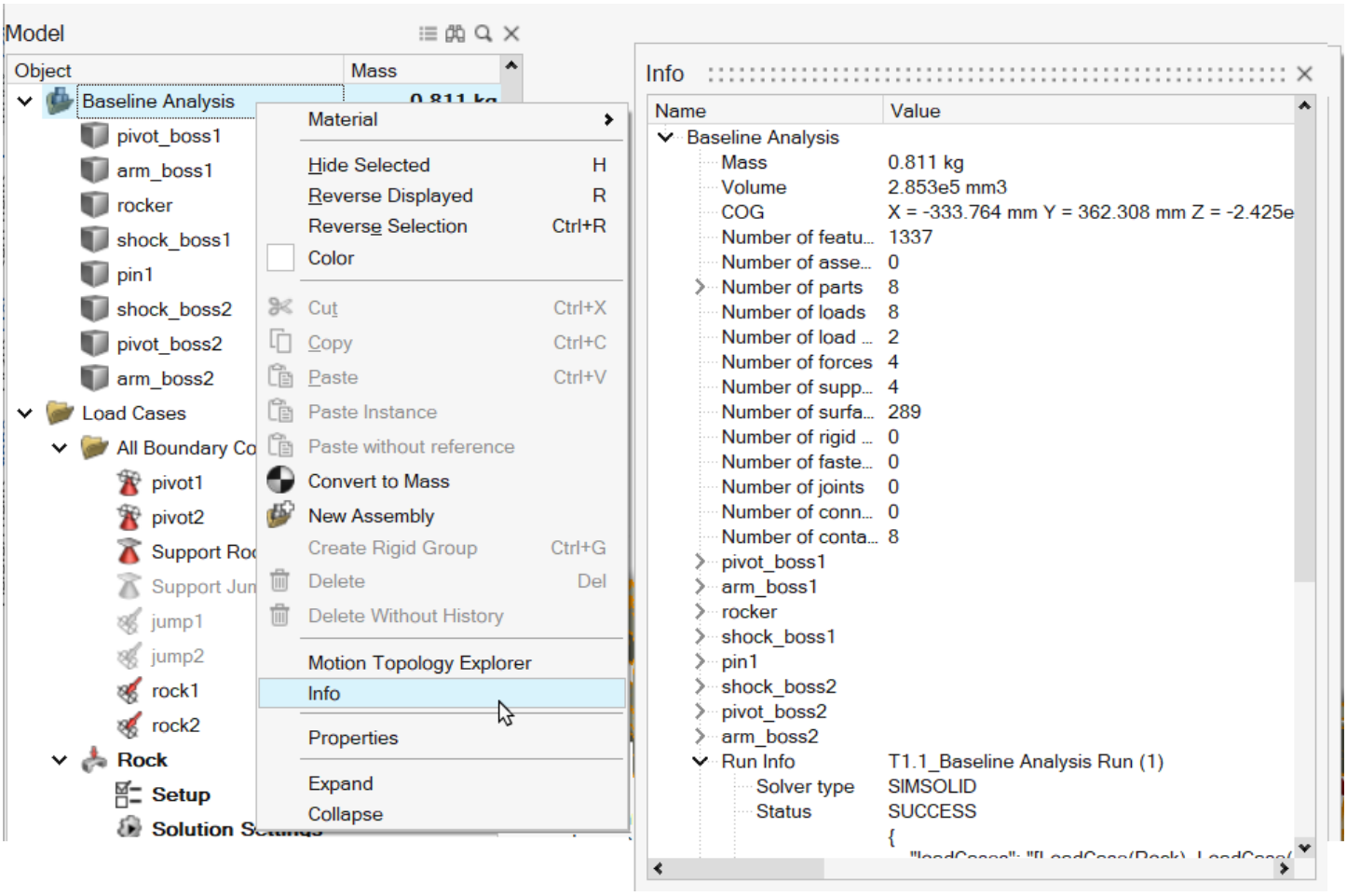
SimSolid Run Status

When running a SimSolid analysis, run progress is displayed in the status bar at the bottom of the Inspire window.



Entity Information

Right-click any entity in the Model Browser and select **Info** to display additional information about that entity.

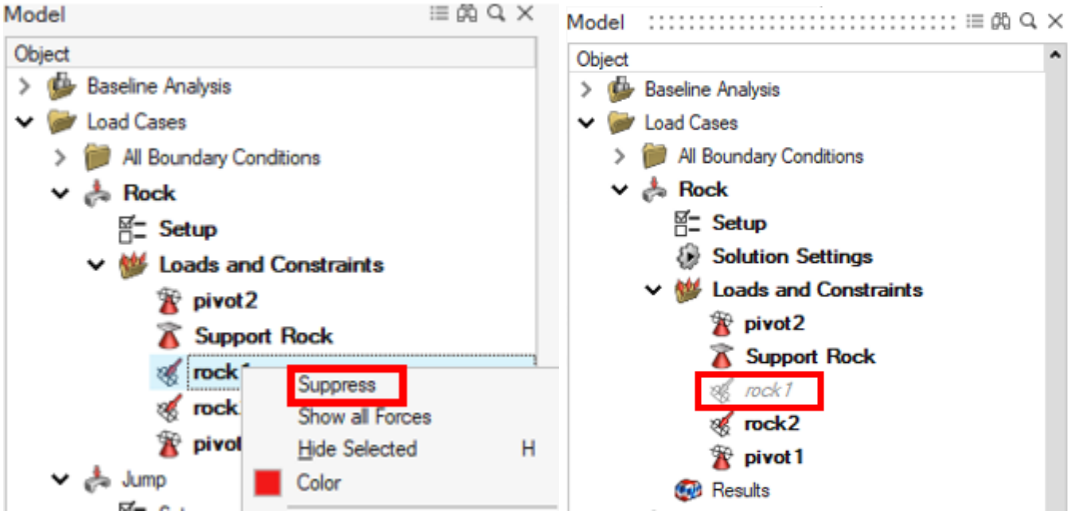


Suppress/Unsuppress Entities

Suppress or unsuppress an entity using the context menu in the Model Browser.

Entities that are suppressed do not appear in the modeling window and are not included in calculations such as optimization, static weight calculations, mass computations, etc.

Entities that are suppressed are displayed in grayed-out, italicized text in the Model Browser and removed from the modeling window until unsuppressed.



For more information, see Suppress or Unsuppress.

Contacts

Preferences > **Inspire** > **Run Options** now allows you to set a default for **Contact type for parts with fasteners**.

Self Contact support was added to the Contacts tool for SimSolid.

For more information, see Contacts.

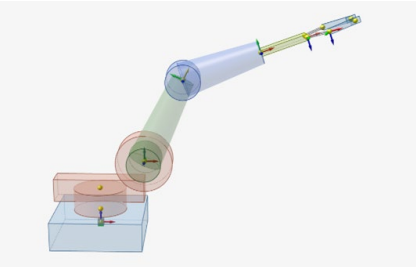
Spot Welds

The spot welds workflow was improved with multiple methods of creation.

For more information, see Spot Welds.

Motion

Multibody simulation can be applied at various design stages, from conceptual design and load/stress calculations to the operational assessment of the overall system. Until now, Inspire Motion has primarily focused on the intermediate stage, where users begin with an existing CAD model and assemble the mechanism based on geometric features to conduct load and stress analysis on critical components.



Initial Stage

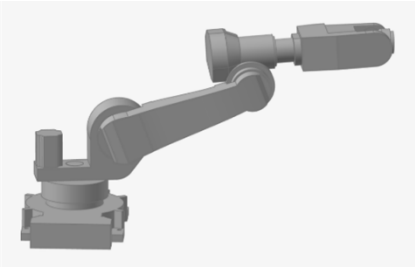
Conceptual Design

Start with a skeleton mechanism to test the general principle.

Outline part shapes and consider interference issues.

Coarsely define actuator characteristics (springs, drivers, etc.) and mass/inertia for each body.

Optimize the design for performance.



Intermediate Stage

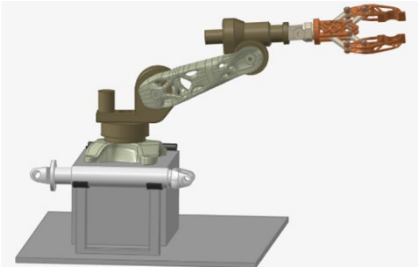
Load & Stress Calculations

Design detailed CAD models and select materials.

Assemble the mechanism based on geometric features.

Perform load & stress analyses on critical components.

Optimize parts for manufacturability.



Final Stage

Operational Assessment

Conduct the most complex analysis to understand real behavior.

Introduce structural compliance and detailed actuator characteristics.

Optimize the overall system behavior.

With this release, Inspire Motion has been enhanced to support the initial design stage, allowing you to start with a skeleton mechanism to define the general principles. At this stage, components often have primitive shapes, and joints are typically based on hard point locations and conceptualized designs.

Furthermore, Inspire Motion has been upgraded with sophisticated entities, enabling the integration of controllers, nonlinear mathematical equations, and multi-physical capabilities by importing Functional Mock-up Units (FMUs). This allows for the simulation of complex systems to assess operational viability during the final stage.

In this release, we introduce approximately 70 new modeling entities into Motion. To maintain a clean interface with easy-to-follow workflows, Inspire Motion now offers two distinct profiles:

Motion Designer Profile: Designers typically begin with CAD, leveraging existing geometrical features to define the connections within their mechanisms. The primary objective is to evaluate the performance and functionality of an existing design. This profile is similar to the previous version of Inspire Motion.

Motion Analyst Profile: Analysts often start with a blank slate, employing a hard-point-based design approach. Hard points represent critical locations, such as mounting points or load-bearing connections within and between assemblies. Using these points, mechanisms are constructed with simplified primitive shapes. Joints, located at the hard points, are conceptual in nature. The focus here is on gaining a fundamental understanding of how a new mechanism might function before applying detailed shapes or refinement.

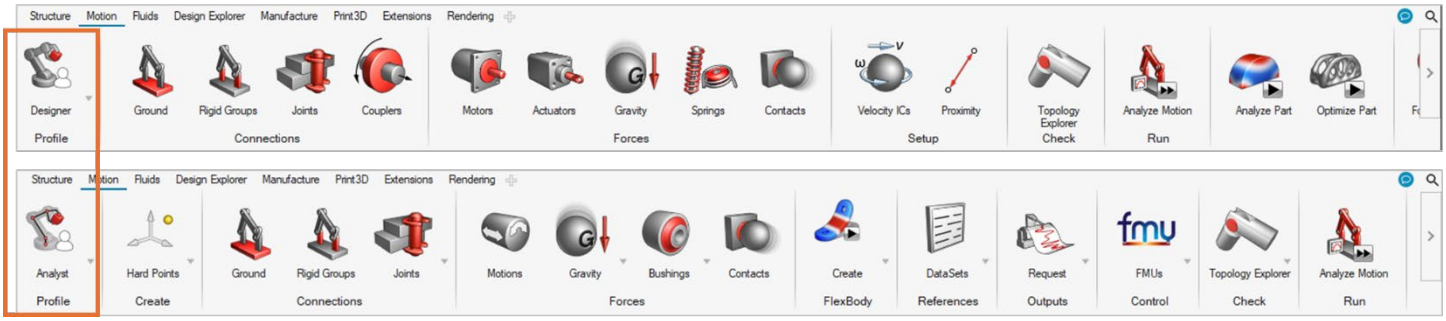
This dual profile approach allows Inspire Motion to cater to both the refinement of existing designs and the conceptual development of new mechanisms, offering flexibility and precision for diverse needs. You can navigate between the profiles.

Motion Analyst

With Motion Analyst being a completely new profile with numerous features, including all details in these release notes is not practical. Please refer to the help for a comprehensive overview of the new features. For more information, see Motion Analyst.

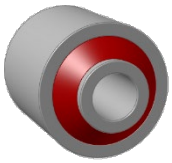
Ribbon Switching (Motion Designer to Motion Analyst)

An icon is available for toggling the main ribbon view between motion profiles.



Bushing Moved to Motion Analyst

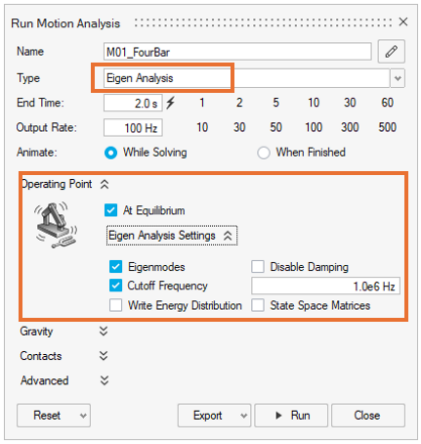
The Bushing feature has been improved and moved to Motion Analyst and is represented by a new icon. Separate from structural bushings, motion bushings now provide additional support for modeling linear and nonlinear behavior as well as symmetrical pairs.



For more information, see Bushings.

Eigenanalysis Moved to Motion Analyst

The Eigenanalysis feature has been moved to Motion Analyst, and like previous versions, can be accessed from the Motion Run Settings.



For more information, see Eigenanalysis.

Flex Bodies Moved to Motion Analyst

Flex body analysis, including Create Flexible Body, have been moved to Motion Analyst under a Flex Body tool group.

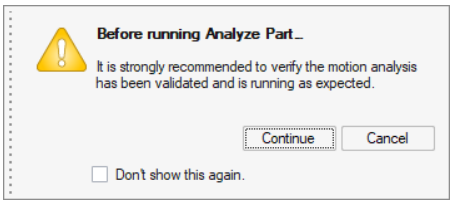
Two additional methods for modeling with flex bodies have been introduced: Replace and Import. With Replace, you can replace any rigid part with a flexible part, using a flex body H3D file. During the replace operation, all joints and motion entities attached to the flex body are preserved. With Import, a flex body H3D can be imported into the model and Motion Analyst hard points are automatically defined at each interface node, ready for connecting joints and other entities to the part.

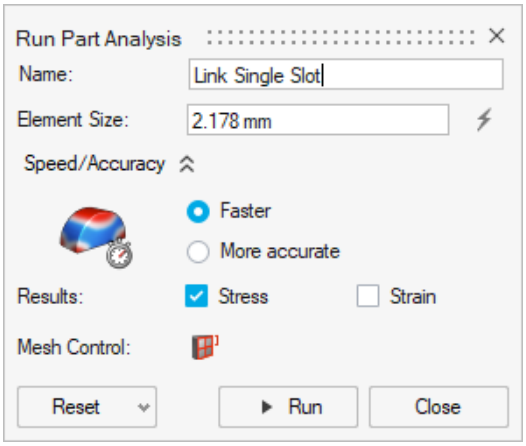
Finally, there is a new View Modes tool that opens the flex body modes review context. Previously, this was accessed by selecting the Create Flexible Body tool.

For more information, see Flexible Bodies.

Improved Analyze Part (Motion Designer)

The Analyze Part feature has been improved to provide faster and more accurate results. Previous methods employed inertia relief FE solutions on a rigid part, using selected load steps from the motion analysis. New methods employ creating an implicit flex body, solving a complete flex body motion analysis and placing you directly into the results review context. Although it is recommended to first verify that the motion analysis is running as expected, there is no longer a need to run the motion analysis as a prerequisite to performing Analyze Part.





For more information, see Analyze Part.

Revised Icons (Motion Designer)

Some icons have been updated with a new look:

Review Group (Motion Designer, Motion Analyst)

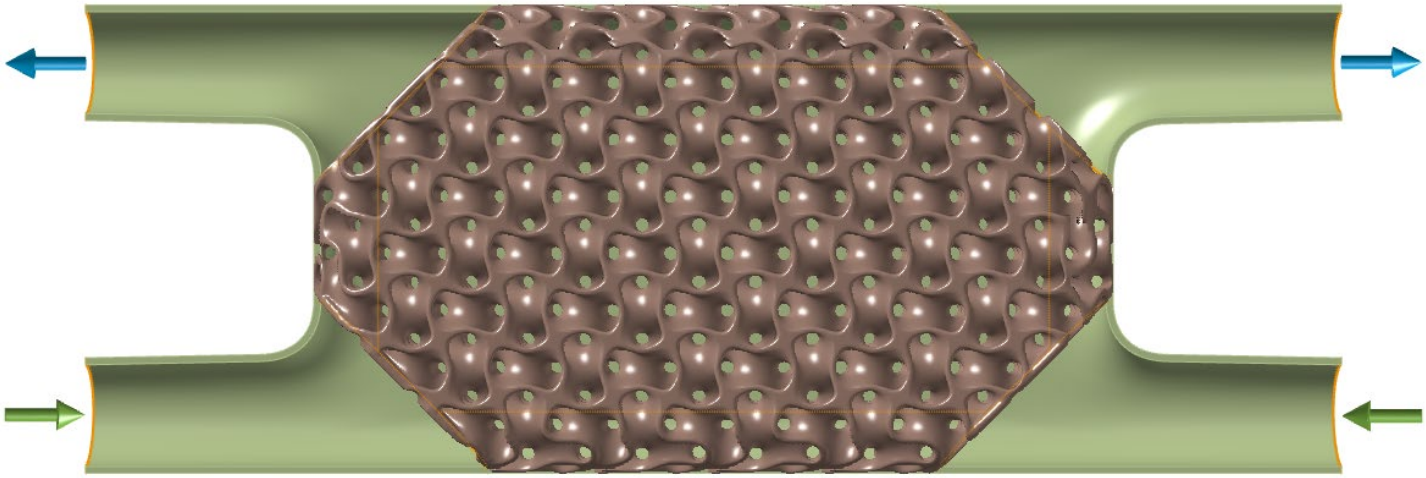
The Review group icons have been rearranged into a dropdown to conserve space on the ribbon.

Fluids

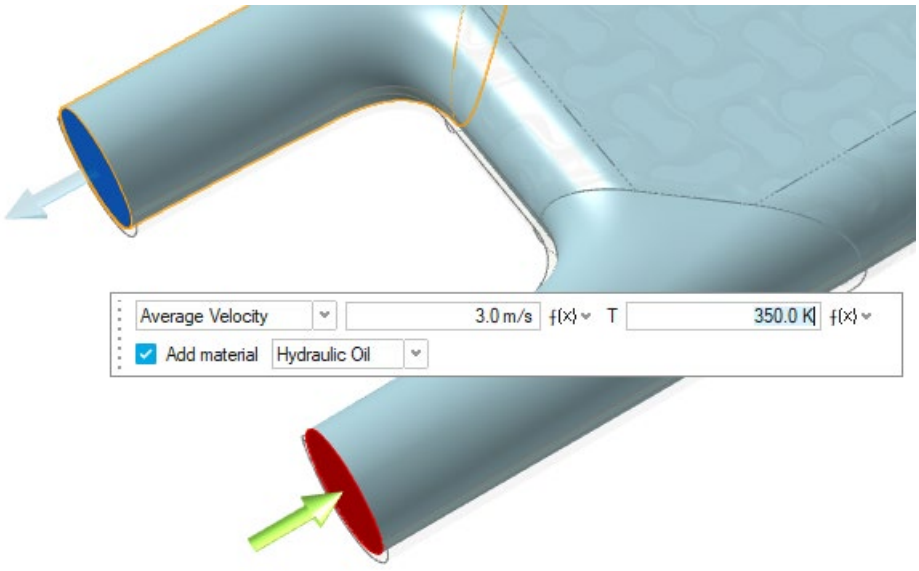
Inlet Material Specification

You can now specify the material that flows through an inlet using the **Add material** option in the inlet microdialog. This is useful when a model has multiple inlets with different nonmixing fluids flowing through them. This feature must not be used for simulating mixing fluids with different materials. One supported application is the Lattice Heat Exchanger, which typically has two different materials entering at two different temperatures. The two fluids are separated by a solid lattice, which prevents the fluids from mixing.

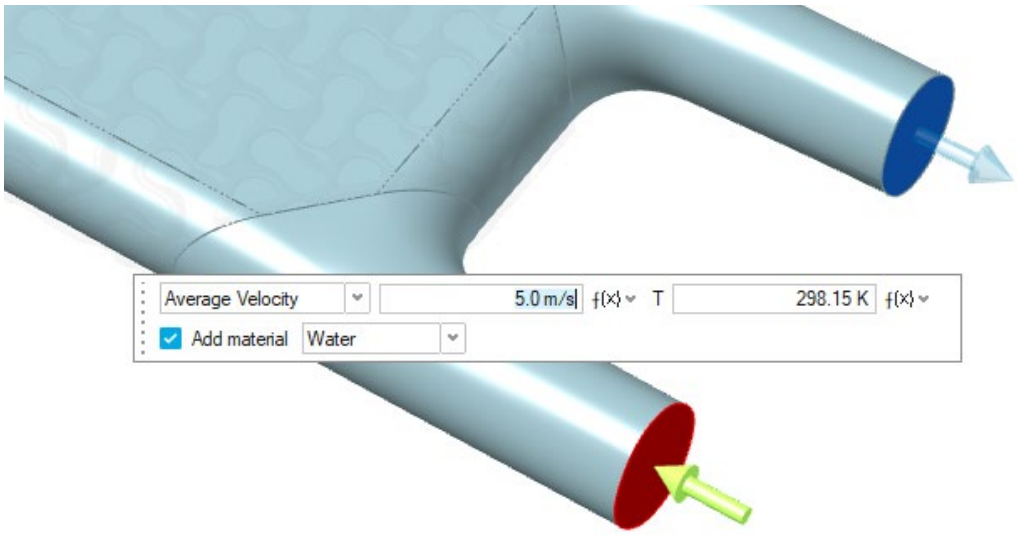
Example Lattice Heat Exchanger with 2 inlets and 2 outlets



Hydraulic oil entering the first inlet at 350 Kelvin



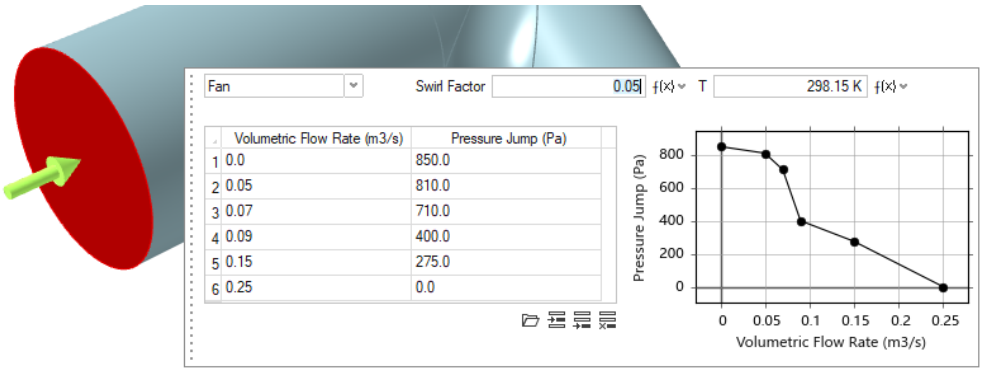
Water entering the second inlet at 298.15 Kelvin



For more information, see Inlets.

Fan Option in Inlet and Outlet Boundary Conditions

The Inlet and Outlet tools now include a **Fan** option to model the effects of a fan on the fluid via the specification of the fan P-Q curve.

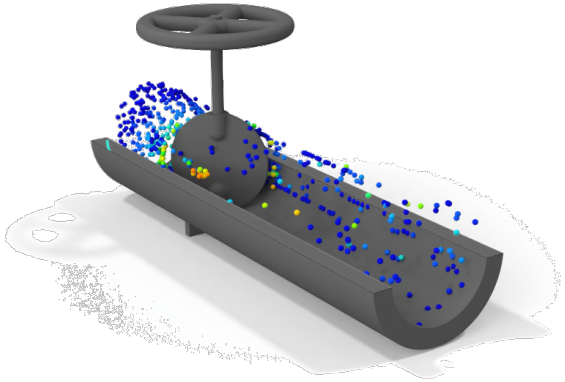


For more information, see Inlets and Outlets.

Particle Visualization

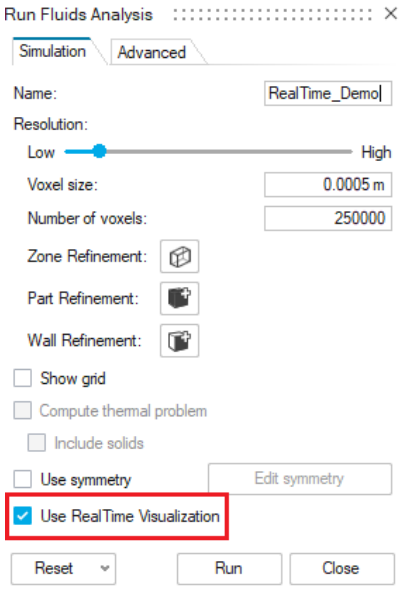
In the Analysis Explorer, you can now opt to view fluids analysis data as moving particles. The particles can be visualized as either spheres or arrows.

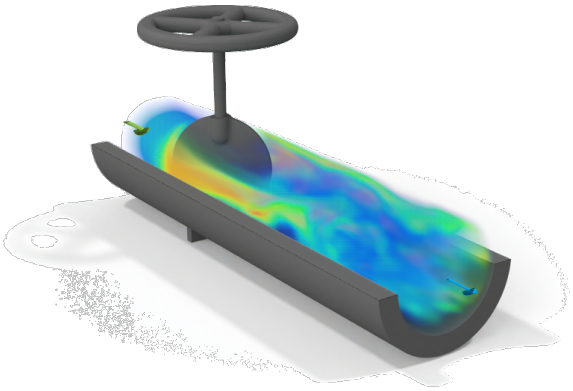
For more information, see Style Options.



Real Time Results Visualization

Results data will appear as they are available rather than after the entire simulation finishes. This feature is supported only when the simulation is performed on NVIDIA GPUs. Currently, volumetric render visualization and section-cuts are supported in real-time mode. Before launching the simulation, select the **Use Realtime Visualization** option in the Run Fluid Analysis dialog to use this feature.

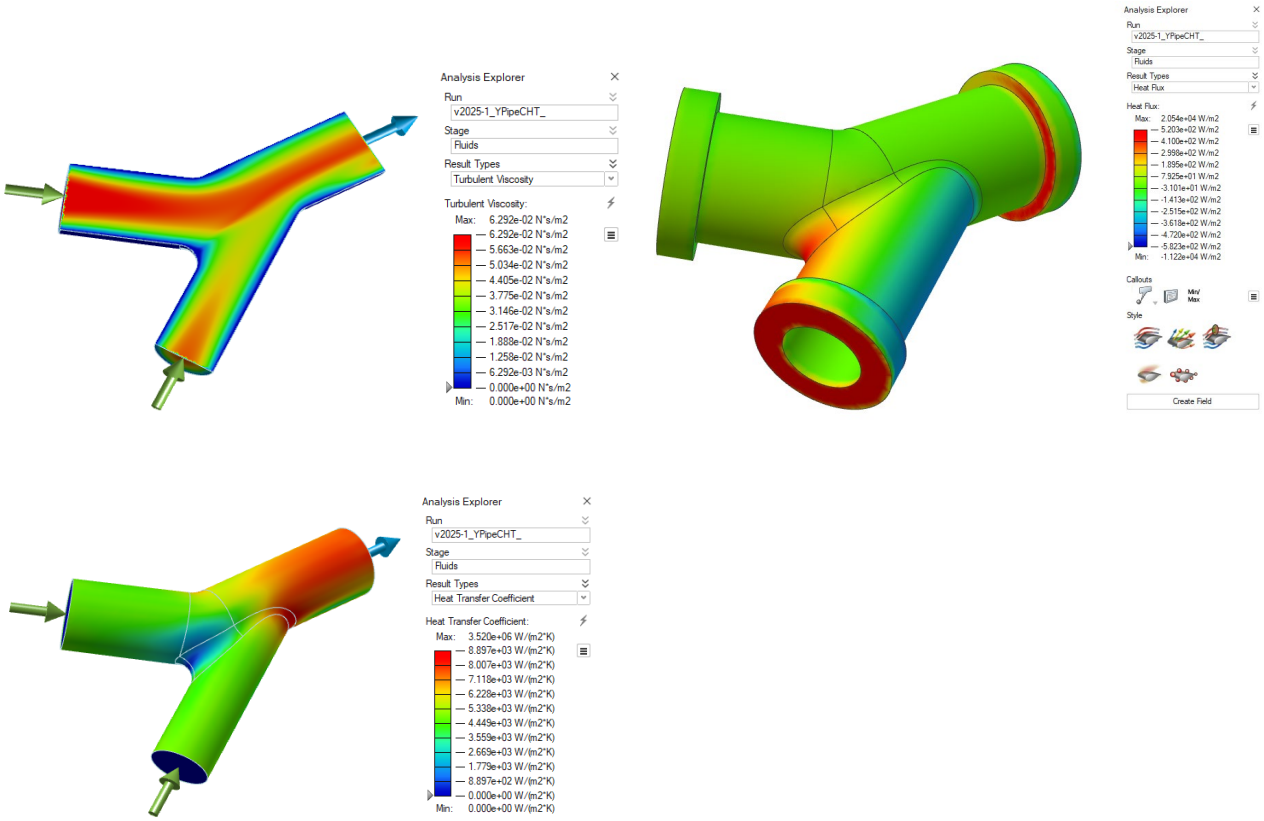




For more information see Run Status.

New Result Types

The following new result types—Turbulent Viscosity, Heat Flux, and Heat Transfer Coefficient—are now available in the Analysis Explorer.



For more information see Result Types.

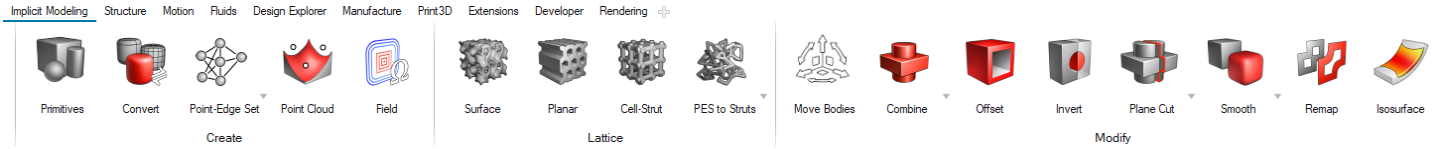
AMD Graphics Cards Support

Fluids simulations can now be performed on AMD graphics cards belonging to the architectures RDNA2, RDNA3, and RDNA3.5.

Implicit Modeling

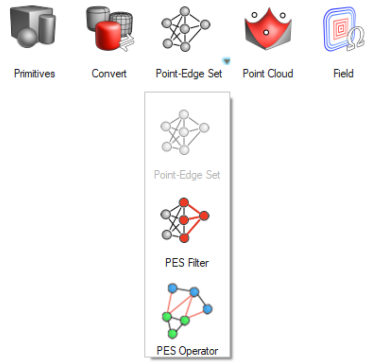
Ribbon Reorganization

The tools on the Implicit Modeling ribbon have been moved into 3 separate groups (Create, Lattice, Modify) to better categorize the tools into fundamental types. The main change is the addition of the Lattice group, moving Convert to the Create group.



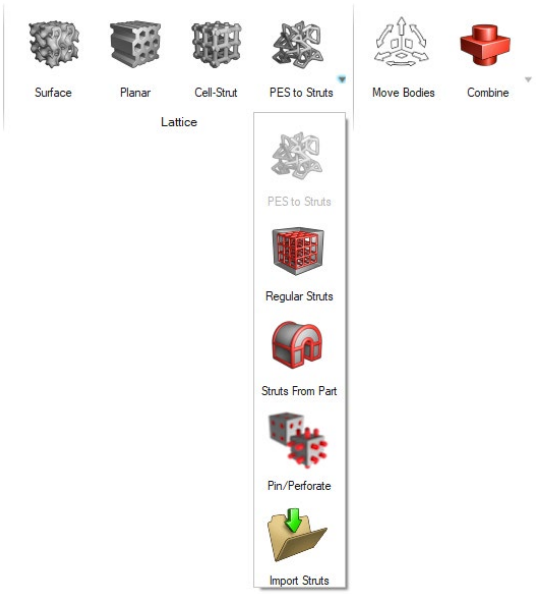
Strut Latticing Overhaul

Point-Edge Set and **Point-Edge Set Filter** are now available through the Implicit Modeling ribbon along with a new **Point-Edge Set Operator** tool.



For more information, see Point-Edge Set.

Point-Edge Sets can be converted into strut lattices using the **Point-Edge Set to Struts** tool (previously **Stochastic Lattice**), and four new modes of strut lattice buttons have been added to the ribbon to streamline common workflows: **Regular Struts**, **Struts From Part**, **Pin/Perforate,** and **Import Struts**.



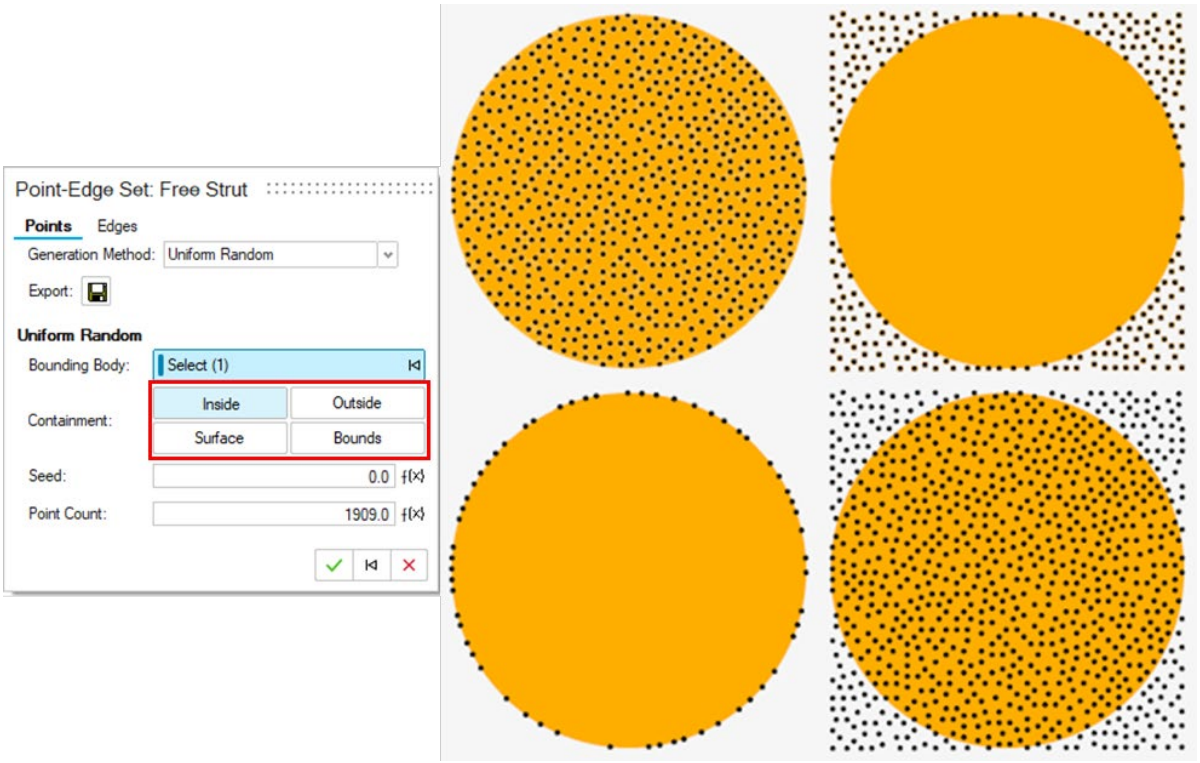
These four buttons will guide you through the process of generating a point-edge set with the correct options and thicken them into struts similar to the workflow of stochastic lattices in 2025.

For more information, see Streamlined Workflows Using Point-Edge Sets.

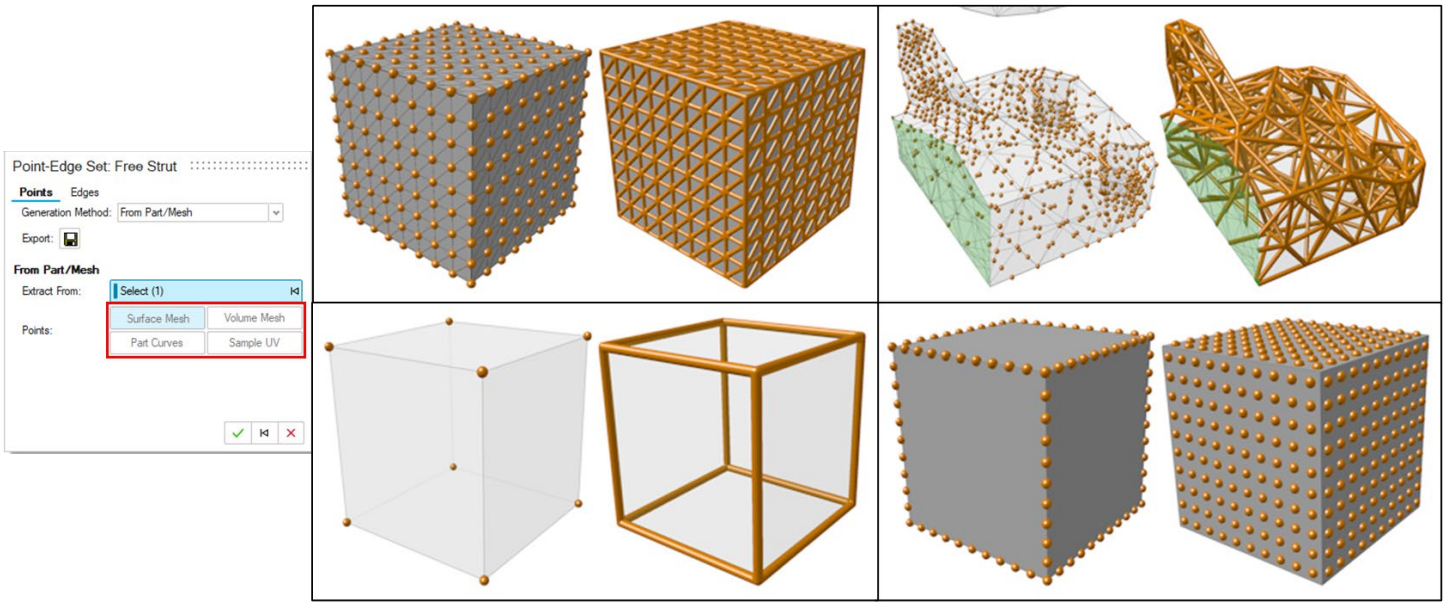
Point-Edge Set Updates

There have been several capability and usability improvements for the **Point-Edge Set** tool.

Uniform Random and **Minimum Spacing** point generation modes now provide four options for generating points: **Inside**, **Outside**, **Surface**, and **Bounds.**

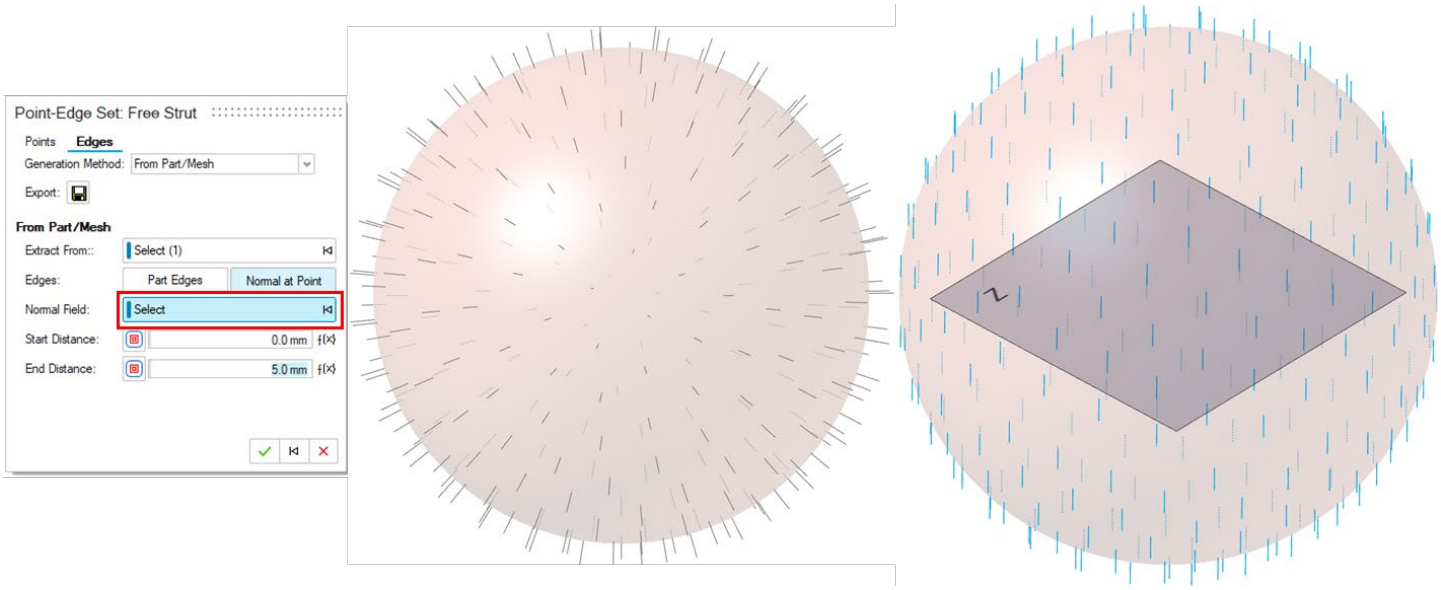


The **From Part/Mesh** generation method now provides four options to extract points and edges from selected objects: **Surface Mesh, Volume Mesh, Part Curves**, and **Sample UV.**

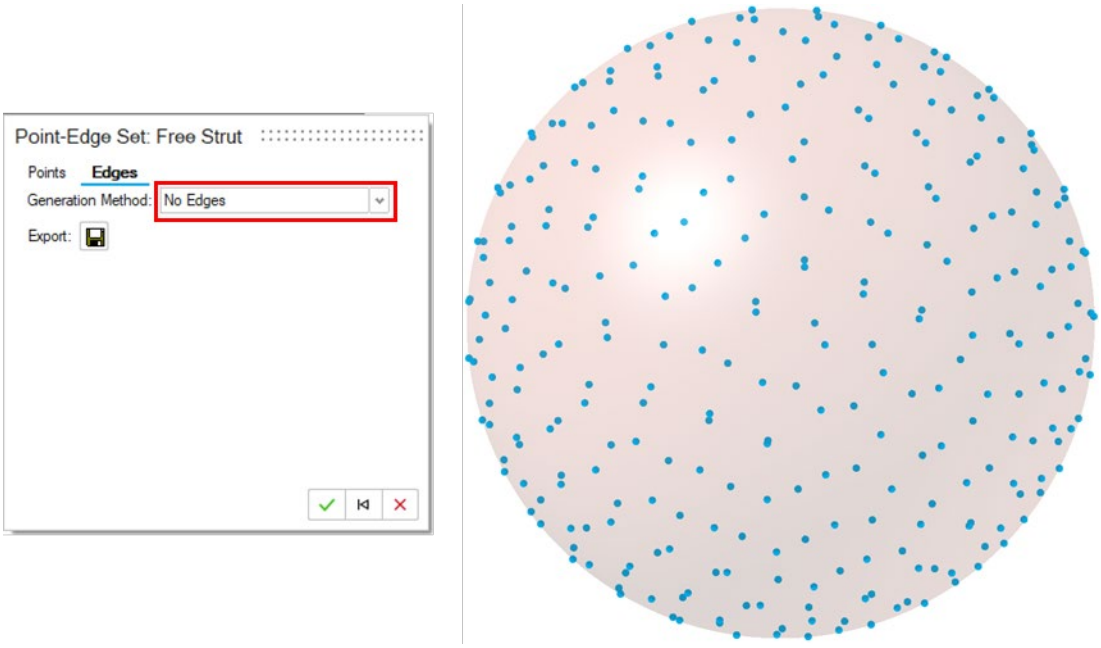


To make it clearer for the user, the buttons will become enabled if that type of data is available on the selected object.

The **Normal At Point** option for creating edges at each generated point now provides the option to provide a custom **Normal Field** as opposed to the target object that was created to generate the points. In the example below, the right image shows what happens when you apply the field of the **Z Plane** to the **Normal Field** parameter as opposed to the default surface normals of the sphere.



It is now possible to generate only points in a point-edge set by selecting the **Points Only** option in the Edge Generation Method. This will skip the edge generation and produce only points.

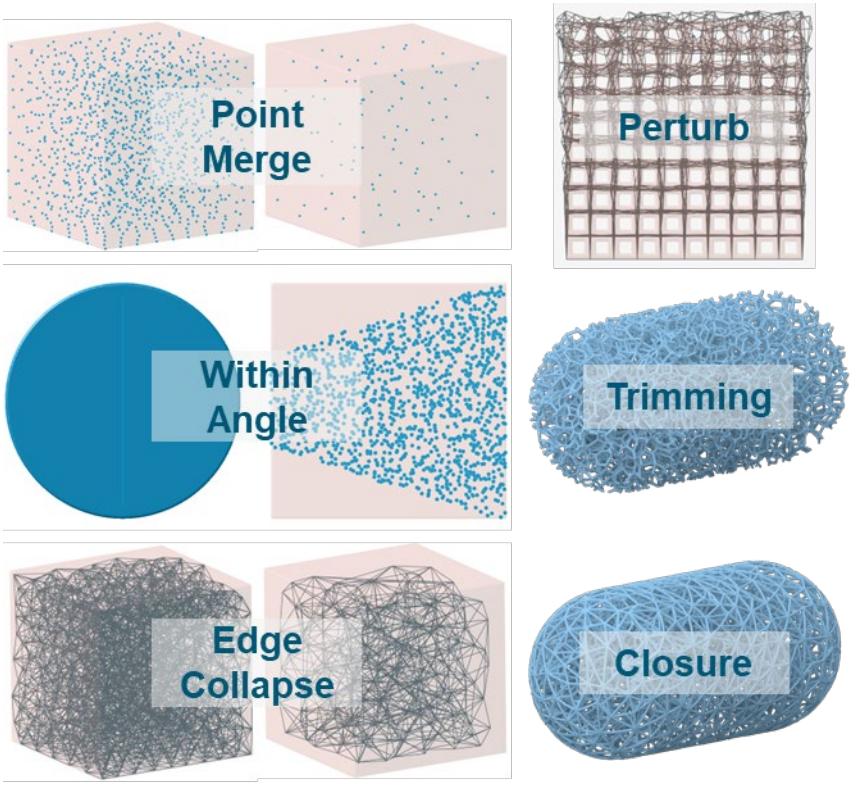


A new **Export** button was added to the Point-Edge Set panel to allow users to save a list of the point coordinates or a list of the edge connectivity to a CSV file in the same format that is used to import point-edge sets and point clouds.

For more information, see Point-Edge Set.

New Point-Edge Set Filters

Four new point-edge filters were added in this release to help fine-tune point-edge sets. Point Merge and Edge Collapse both help reduce the density of points/edges in the set. A new Point Angle filter keeps/removes points based the angle of a target field and a reference field at each point. Points can now be shifted either by a specific or random amount by using the Perturb point filter. In 2025 the Trimming filter was added, but in 2025.1 it is now possible to close the lattice ends that were trimmed by the filter, either connected to nearest points directly or across the surface.



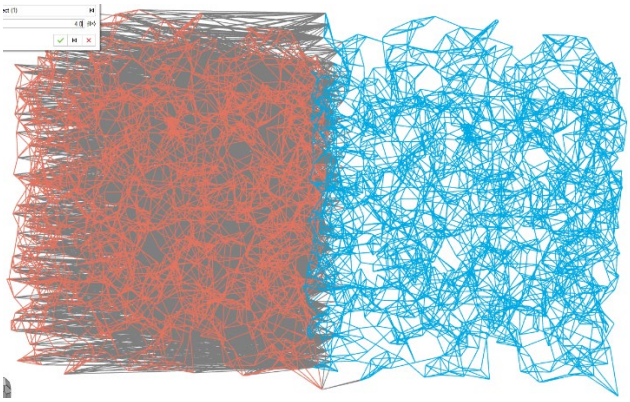
For more information, see Point-Edge Set Filter.

Point-Edge Set Operator

A new Point-Edge Set Operator tool has been added to allow operations between two or more point-edge sets. In the 2025.1 release there are two operators to use: Union and Bridge.

Union will simply combine multiple point-edge sets into a single point-edge set similar to a Boolean combine. However, any duplicate points or edges will be removed, and only unique points and edges will be preserved.

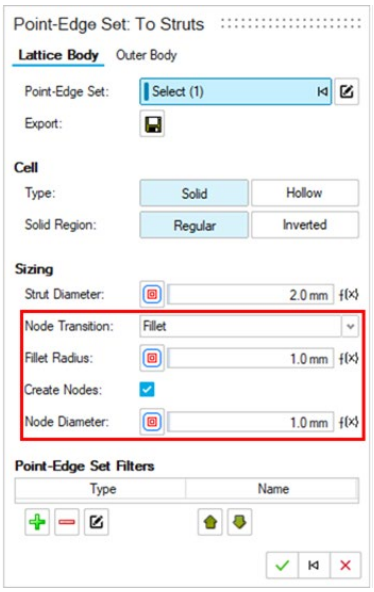
The bridge operator will connect points from the target set to the nearest N points in the tool set, which is specified with the **Valence** parameter.

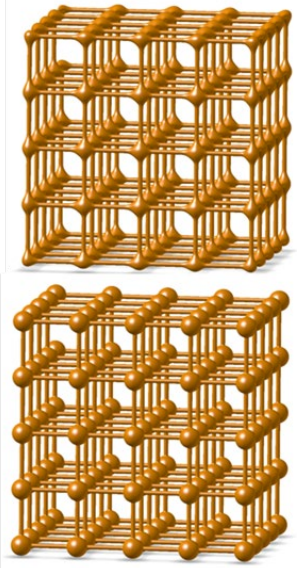


For more information, see Point-Edge Set Operator.

Strut Fillets and Nodes

The Point-Edge Set To Strut tool now provides the option to create spheres at each node in the point-edge set, as well as add fillets or chamfers at the intersection of struts.

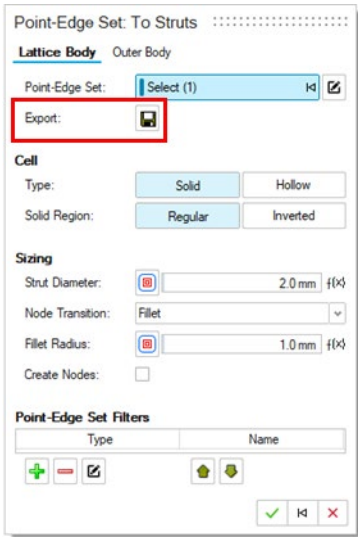




For more information, see Streamlined Workflows Using Point-Edge Sets.

FEM Export of Struts

Strut lattices can be directly exported to an FEM file from the Point-Edge Set to Struts tool using the new **Export** button.

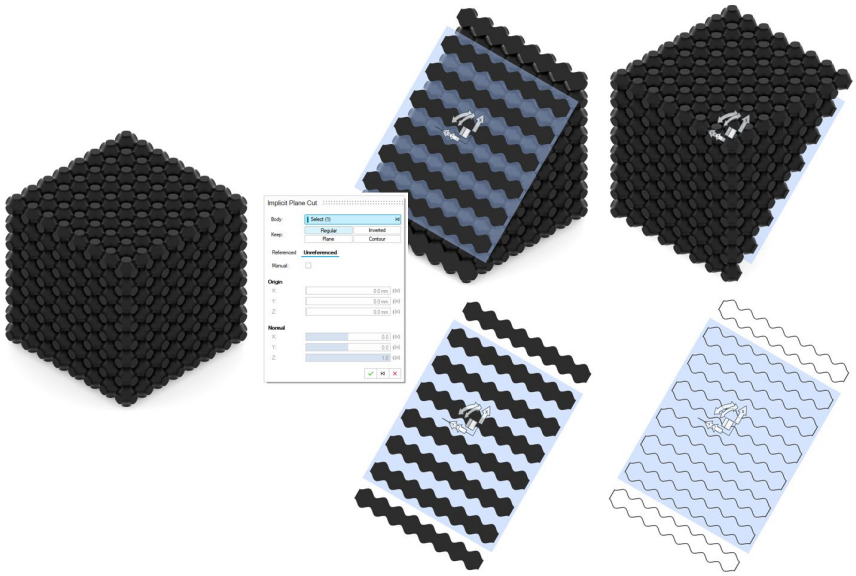


This file format can be used in various Altair solvers and removes the need for exporting the lattice geometry as a mesh. The struts are exported as CBAR elements with PBAR properties for the thicknesses of the struts. 1D elements can also be imported into Inspire through an FEM file and converted to a point-edge set through the From Part/Mesh mode.

For more information, see Streamlined Workflows Using Point-Edge Sets.

Plane Cut

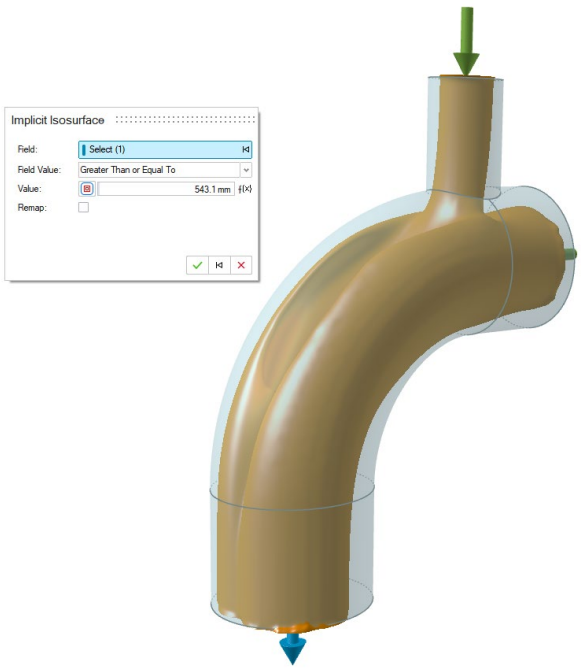
A new Plane Cut tool has been added to the Implicit Modeling ribbon to allow cutting an implicit body with a manually defined or dynamically linked plane. There are four modes to the plane cut tool: **Regular**, **Inverted**, **Plane**, and **Contour**. These modes can be seen below:



For more information, see Plane Cut Implicit Geometry.

Isosurface

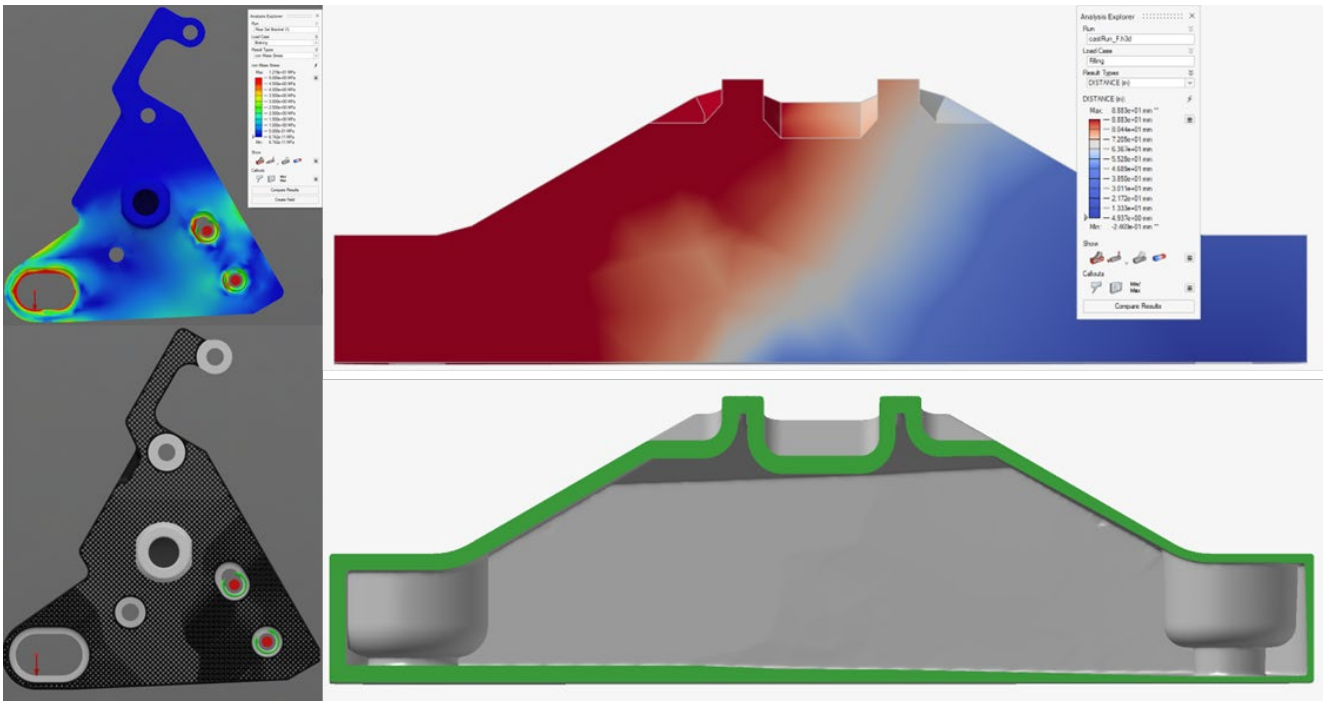
An Isosurface tool has been added to the Implicit Modeling ribbon to allow extracting surfaces from any type of implicit body or field. This is especially useful to create a surface from analysis results and create optimized topologies for results from solvers other than structural analysis. For example, the tool can be used to create a more optimal fluid volume based on Inspire Fluids results.



For more information, see Isosurface Implicit Fields.

Fields from More Analysis Results

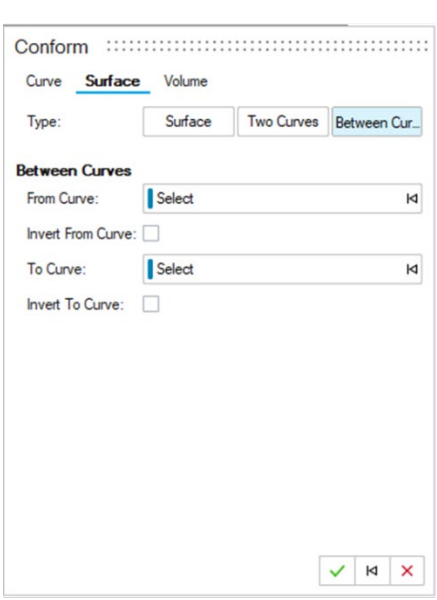
In 2025, Inspire added the ability to create implicit fields directly from the Analysis Explorer for OptiStruct and Fluids results. This has been expanded to all results types that are attached to a solid body, including imported analysis data through H3D files. For example, on the left, a field was created from SimSolid analysis results; on the right a field was created from Inspire Cast results through an imported H3D file (the shell thickness is based on the distance to the ingate in the casting analysis).

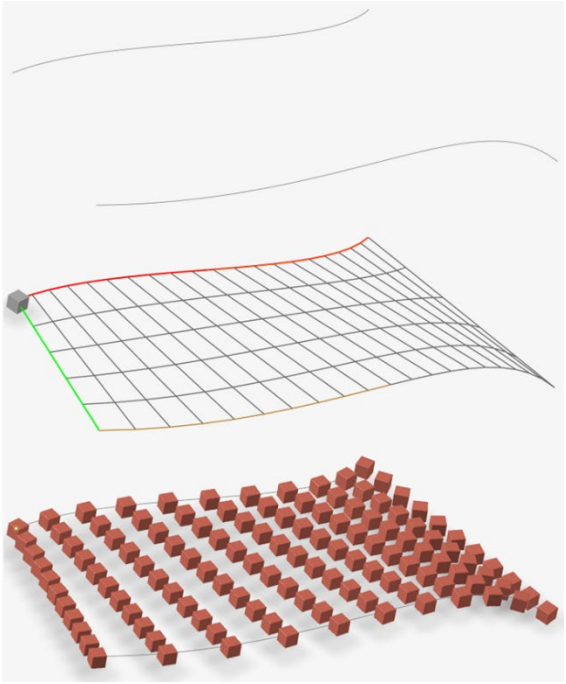


For more information, see Create Implicit Fields from Analysis Results.

Curve-To-Curve Conformal Pattern

A new option has been added to the surface tab of the conform panel which allows you to pick two curves and create a lofted UV space between the two curves.

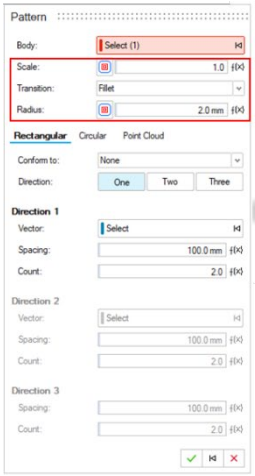


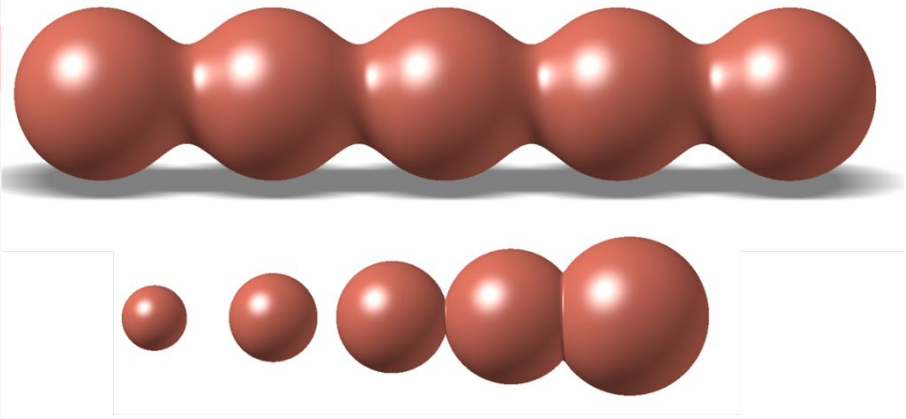


For more information, see Conformal Coordinate Spaces.

Pattern Smoothing & Scaling

Fillets or chamfers can now be added between copies in an Implicit pattern, as well as a scaling factor for patterned objects. Both parameters can be field driven.

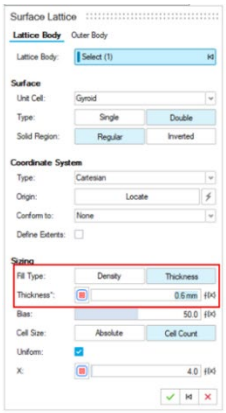


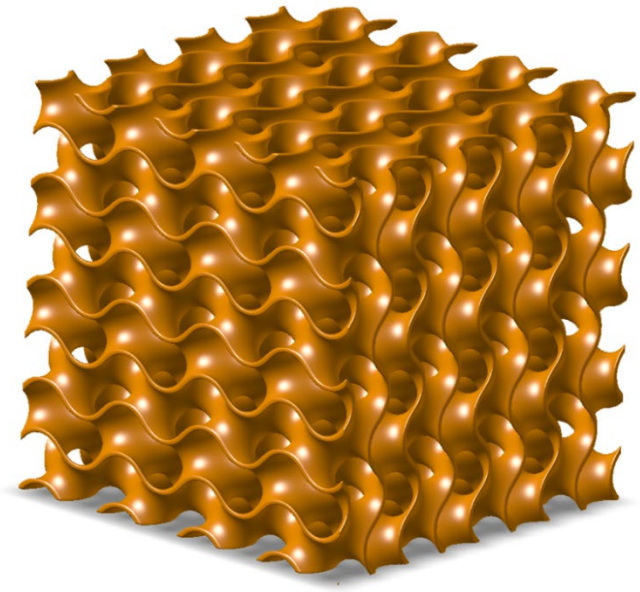


For more information, see Pattern Implicit Geometry.

Thickness Parameter for Double Surface Lattices

You can now choose between specifying a density or an actual thickness value. The thickness value can be constant, variable, or field driven. This is a very important parameter to control exact sizing for components such as heat exchangers. NOTE: Due to the nonlinear nature of surface lattice equations, the thickness is technically an approximate value. The thickness should be very accurate near the non-thickened surface and may lose tolerance farther away from the non-thickened surface.

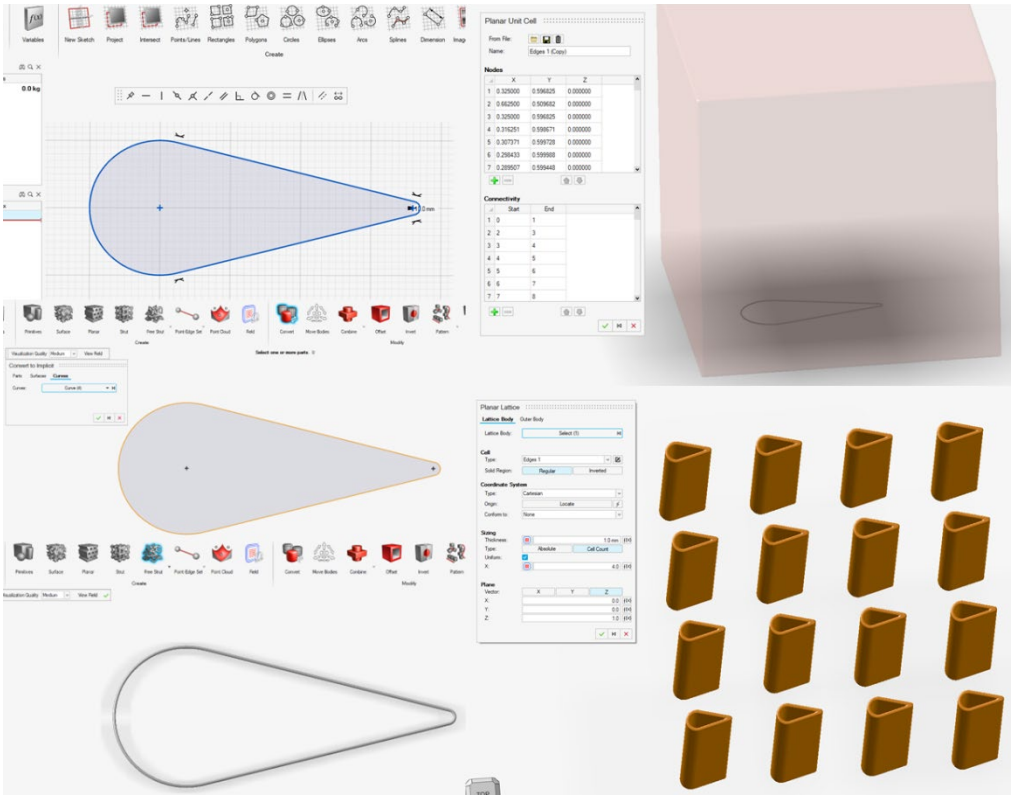




For more information, see Create an Implicit Surface Lattice.

Dynamic Link to Sketches

Sketches can be converted directly in the Implicit Convert tool, and all curves contained within the sketch will be converted to implicit curves. These can be used in various workflows, either in a point-edge set using the From Part/Mesh mode, used directly in a unit cell for Planar or Cell-Strut lattices (it will be listed in the **Unit Cell** dropdown), in a field, or directly in any Modify tool such as Offset. Any edits made to the sketch will be updated in the implicit part, just like any other dynamic link from converted parts.



For more information, see Convert to Implicit Geometry.

Cross-Implicit Part Referencing

You can now select implicit parts later in the construction history to reference implicit bodies from earlier implicit parts. A dynamic link will be maintained across the two parts, so if the previous implicit body is edited, this will be reflected in the later part. It is recommended to hide the body in the first part as it will look duplicated in the second part.

For more information, see Construction History.

Enhanced 3MF Extension Support

The support for the 3MF beam extension has been enhanced to also support per-node radii both for import and export. Imported 3MF files with beam extension data will now be directly loaded in as an implicit point-edge set and thickened to the appropriate radii contained in the 3MF data.

3MF files from Inspire can now include volumetric data as well as mesh and beam data. The volumetric data resolution will match the visualization quality of the implicit parts in the model. For now, importing 3MF volumetric data is not supported.

For more information, see Supported File Formats.

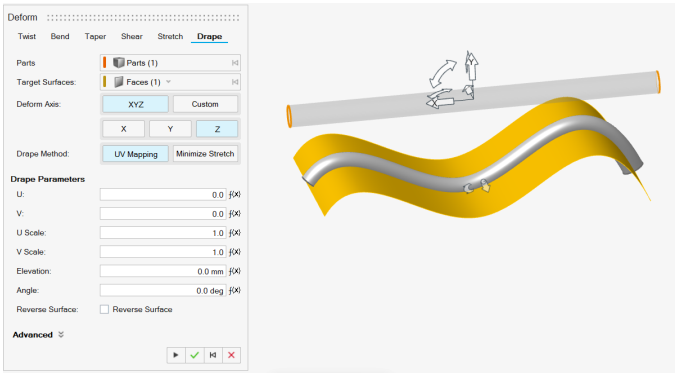
Performance Improvements

The implicit engine architecture was significantly overhauled to avoid unnecessary recomputing of geometry (implicit construction history). You should expect a general widespread speedup of implicit modeling workflows.

Geometry

Drape

The Deform tool now includes a Drape tab that you can use to drape one or more parts over a surface.



For more information, see Deform.

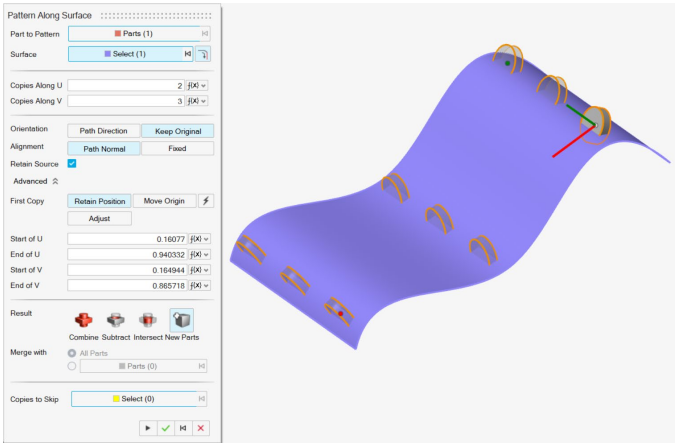
Custom Deform Axis

The Deform tool now includes a **Custom** setting in each tab, allowing you to deform parts with respect to a linear edge or axis selected using the **Axis** collector.

For more information, see Deform.

Pattern Along Surface

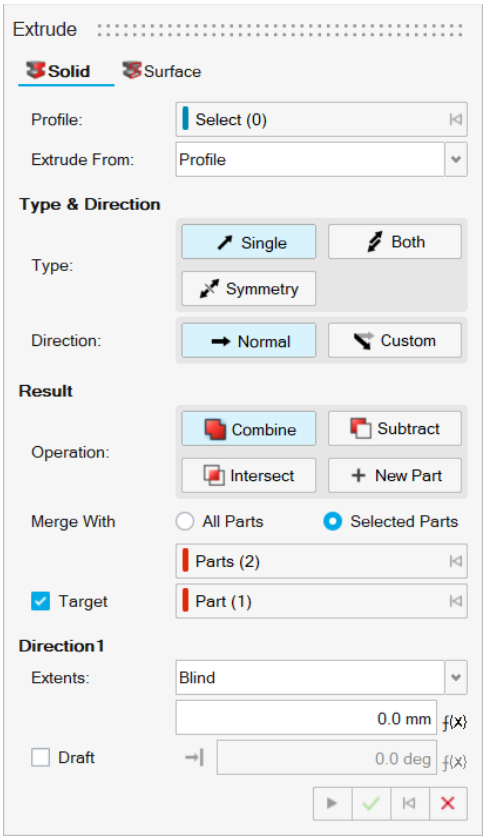
The new Pattern Along Surface tool allows you to select parts and create copies over a surface.



For more information, see Pattern.

Target Part for Combine Operations

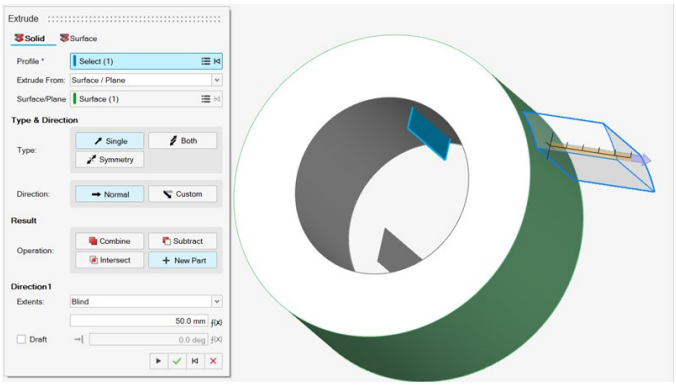
In the Extrude, Revolve, Sweep, Multi Sweep, and Loft tools, you can use the **Target** checkbox and collector to select the target part when performing a Combine operation.

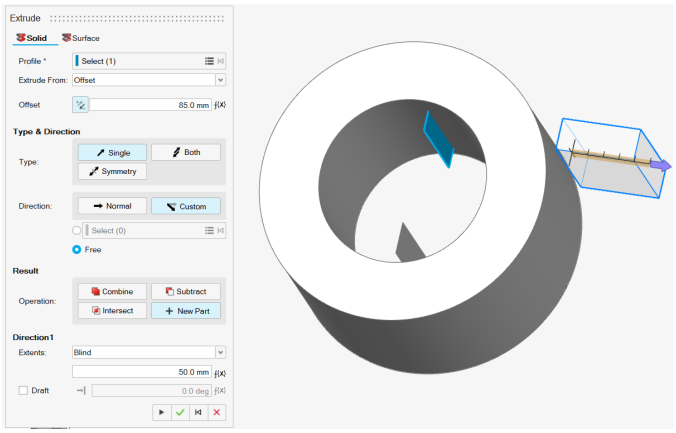


For more information, see Extrude. Revolve, Sweep, Multi Sweep, or Loft.

Updated Extrude Tool

You can now use the Extrude tool to extrude from a surface or plane, or to extrude a profile with an offset:





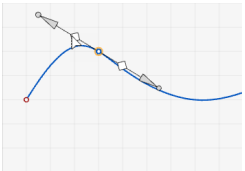
For more information, see Extrude.

Sketching

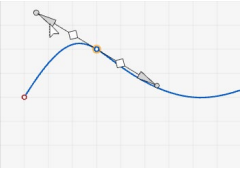
Spline Tangent and Magnitude Handles

When using the **Spline Through Points** tool, handles are displayed to allow you to change the tangency and magnitude of the spline curve.

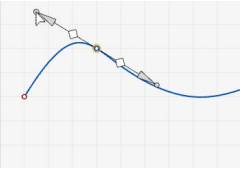
Drag the diamond control to adjust the tangent curve:



Drag from the arrowhead at either end of the tangent line to change the magnitude:



Drag the point at either end of the tangent line to change tangency and magnitude simultaneously:



For more information, see Splines.

Show Image Plane Outside of Sketch

The Image Plane feature now includes a **Show Image Plane Outside of Sketch** option.

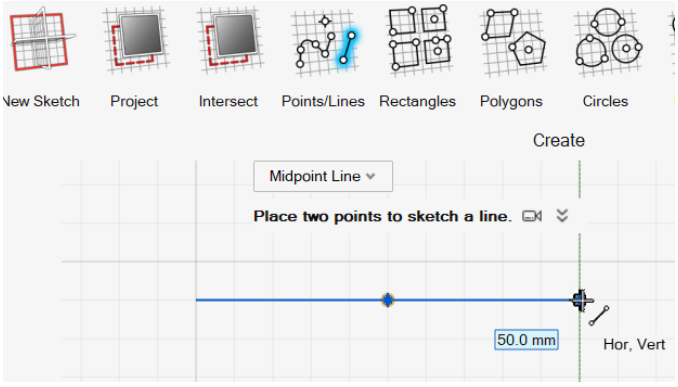
If you want the image to remain visible outside of the Sketch context, right-click the modeling window and select **Show Image Plane Outside of Sketch** (or **Preferences > Sketching > Show Image Plane Outside of Sketch**). When the option is not selected, the image plane is not visible when you leave the Sketch context.

For more information, see Image Plane.

Sketch Midpoint Line

The Line tool now includes a dropdown to choose the type of line to create.

Choose **Midpoint Line** to create a line using the midpoint and either endpoint.



Choose **Single Line** to create a line by defining the end points.

For more information, see Points/Lines.

Convert to Construction and Convert to Infinite Lines

The Sketch Constraints toolbar now includes options to convert a sketch curve to construction geometry and convert a line to an infinite line.

For more information, see Sketch Constraints.

Hide Sketch Plane After Sketch Creation

Use **Preferences** > **Sketching** > **Hide Sketch Plane After Sketch Creation** to control whether the reference plane is hidden or visible after exiting the Sketch context.

For more information, see Preferences: Sketching.

Picking Priority

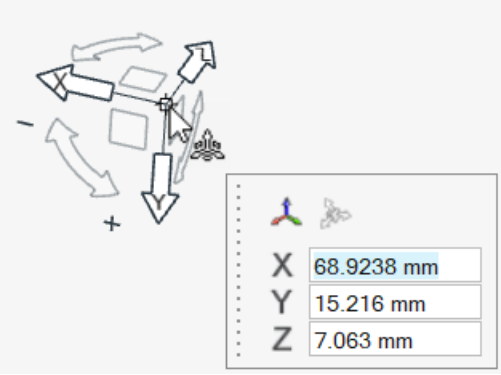
Picking priority is now given to underdefined sketch entities if they overlap fully defined entities.

PolyNURBS

Move Bodies

When you use the Move Bodies tool, the last-used alignment is now saved. In previous releases, the axis was aligned to the global axis.

To switch between the global axis and PolyNURBS alignment, click the center of the graphic manipulator:

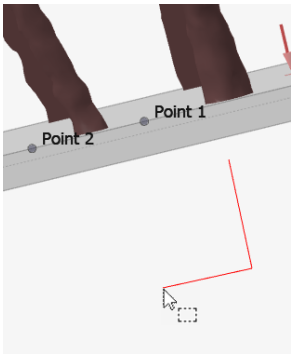


For more information, see Move Bodies.

Pave

The Pave tool includes the following improvements:

- Reference points are now supported when using the Pave tool.
- You can Ctrl-click in space with the Pave tool to use the nearest global plane.



Ctrl-click a reference axis or plane to select it. While the reference is selected, all points you click will snap to the axis or plane. Ctrl-click the reference again to deselect it.

For more information, see Pave.

Rendering

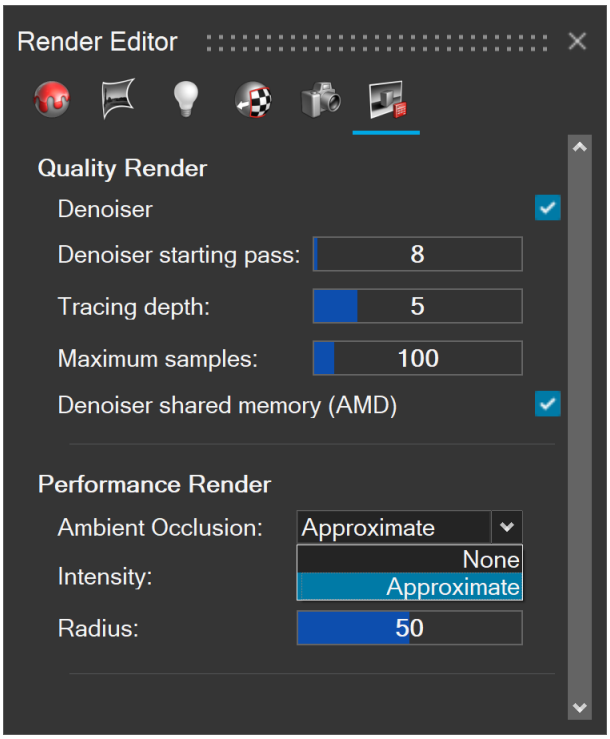
Blend

The new Blend option of the Texture Positioning tool allows you to control intersecting textures to either blended or sharp transitions.

Ambient Occlusion

Performance Render mode now has options to control lighting and shadow effects. **Intensity** and **Radius** parameters allow you to increase or decrease the ambient occlusion effects.

Performance Render options in the Settings tab of the Render Editor



Ambient Occlusion – None

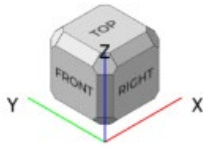
Ambient Occlusion – Approximate 100%

For more information, see Define the Rendering Settings.

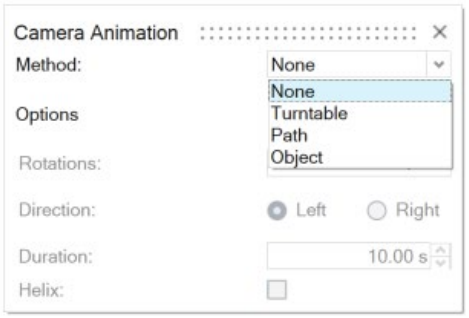
General

Camera Animation

You can now animate the camera to view your model and record the animation in any display mode. For models without animated results, in the view controls in the lower left of the modeling window, click the **Create Camera Animations** tool to activate the timeline. (For models that already include animated results, the timeline appears automatically.)



Then, click the icon at the far right of the timeline to access **Turntable**, **Path**, and **Object** animation options.



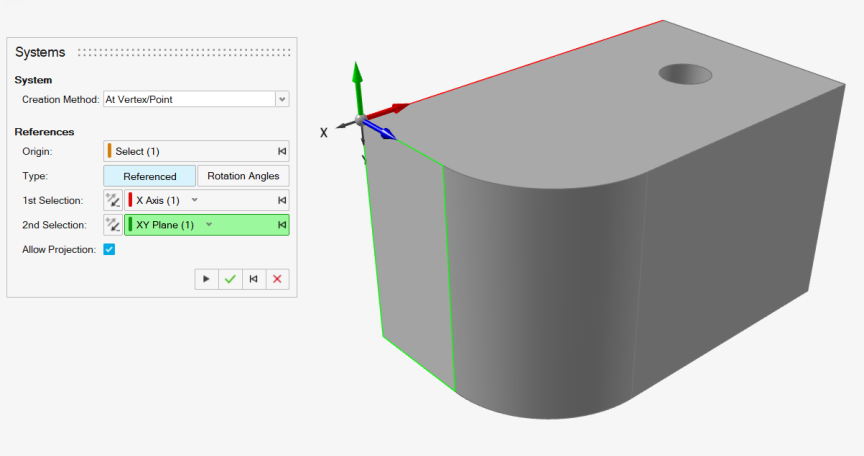
- **Turntable** lets you set rotation direction, number of rotations, duration, and enable **Helix** for vertical movement.
- **Path** allows you to define multiple camera positions and set the duration between them.
- **Object** lets you choose a part for the camera to stay aligned with.

For more information, see Create Camera Animations.

Updated Systems Tool

The Systems tool has been moved to the Home tools.

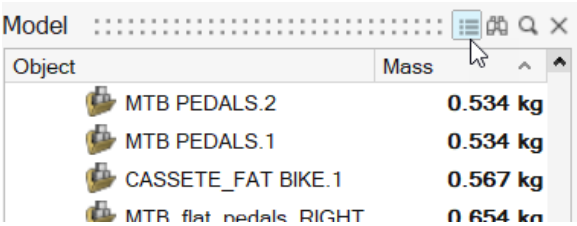
You now use it to create and apply user-defined coordinate systems to loads, supports, and displacements. This is useful for aligning loads to a local coordinate system rather than the global coordinate system. Keep in mind that results are only shown in the global coordinate system.



For more information, see Systems.

List View in Model Browser

Select **List View** to view the Model Browser as a sortable list view with groups of assemblies and parts. You can click a column heading to sort each group in increasing or decreasing order.

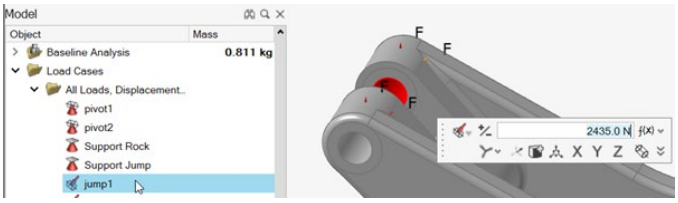


When **List View** is not selected, the Model Browser is displayed in a hierarchical tree view.

For more information, see Change the Model Browser View.

Double-Click to Edit Entities

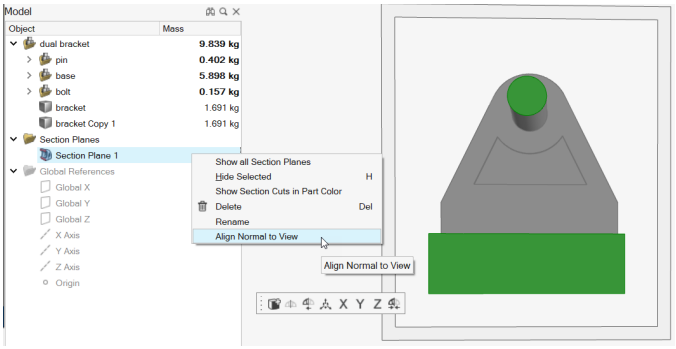
Double-click an entity in the Model Browser to edit it.



For more information, see Edit.

Align Section Cut

To align the model so the section cut is aligned with the current view, right-click the section plane in the Model Browser and select **Align Normal to View**.



For more information, see Create a Section Cut.

Collapse/Expand Ribbon

To switch between having the ribbon collapsed or expanded, do one of the following:

- Go to **View > Ribbons > Collapse Ribbon**.
- Double-click a ribbon tab to collapse or expand the ribbon.

For more information, see Workspace.

Guide Panel Redesign

Guide panels for all geometry tools have been redesigned.

Python API

Geometry

- Added support for Pattern Along Surface to create copies of parts over a surface.
- Added support for Drape capabilities to drape one or more parts over a surface.
- Added Target Part for Combine Operations capabilities to Extrude, Revolve, Sweep, Multi Sweep, and Loft tools, allowing you to set the target during combine operations.
- Added support for creating parametric coordinate systems. Use the API to create and apply user-defined coordinate systems to loads, supports, and displacements.

Implicit Modeling

- Added support for creating point-edge sets, point-edge set filters, and point-edge set operators.
- Added support for Plane Cut to cut implicit bodies with four different modes such as Regular, Inverted, Plane, and Contour.
- Added Isosurface capabilities to extract surfaces from any type of implicit body or field.
- Enhanced Pattern with Fillet, Chamfer, and Scale options to add fillets or chamfers and scale to the copies of implicit patterned objects.
- Enhanced Surface Lattice with a Thickness option to control the exact size of the components.

Structure

- Added support for creating and modifying Structural Linear Static, Buckling, Modal, and Modal Prestress load cases to control the buckling, modal, and modal prestress at the load-case level.
- Added support for creating and modifying solution settings to control the SimSolid solution settings such as Solution Adaption, Groups, Adapt to Feature, Adapt to Thin Solids, and Refinement at the load-case level.
- Added support for creating and modifying pretension groups to add pretension groups and control the pretension force per individual fasteners.

Enhancements

- Enhanced the getReactionForces to return reaction forces and reaction moment's components along with position. [INSPIRE-46966]
- Supported parametric variable on Solid Ellipsoid primitive.
- Added support for Sketch Midpoint Line; the Line tool now includes a dropdown list to choose the type of line to create.
- Enhanced Seam Weld capabilities
- Enhanced geometry Tag capabilities
- Enhanced the Extrude capabilities; you can extrude from a surface or plane or extrude a profile with an offset.

DEPRECATED FEATURES

The following features and options have been deprecated in Inspire 2025.1:

- The Solver Expression input option found in the Profile Editor has been removed from Motion Designer. In exchange, Motion Analyst supports a wider array of options for writing custom input expressions for motions and forces. Existing expressions are supported.

Collaboration Tools

- The Collaborations tools have been deprecated in this release.

RESOLVED ISSUES

- Fixed an issue that caused some older models to crash when loaded into a newer version of Inspire. [INSPIRE-47615]
- Fixed an issue that prevented setting the default Inspire product version to launch and load an STMOD file from Windows Explorer. [INSPIRE-47465]
- Fixed an issue where the IPS units were not respected in the material properties and reaction forces when generating a report in Inspire. [INSPIRE-47463]

KNOWN ISSUES

- When exporting a load case to a CSV file, the following parameters are not exported. [INSPIRE-47918]
    - Load case types (Modal)
    - Linked Load cases (Modal Prestress, Buckling)
    - Number of Modes
    - Grounded Fasteners
    - Pretension
    - Inertia Relief
    - Connectors.
- When using an AMD GPU, recent diver updates can cause issues with rendering and implicit visualization. Rolling back to a previous AMD driver version can resolve the issue.
- When running a Fluids analysis and viewing **Heat Transfer Coefficient** and **Heat Flux** result types in the Analysis Explorer, vectors are displayed in grayscale and can cause the application to crash.
- When running a Fluids analysis, **Heat Transfer Coefficient** and **Heat Flux** result types are not hidden as expected when the **Static Streamlines** and **Vectors** options in the Analysis Explorer are selected.
- When running a Fluids Analysis on Linux, the fluid velocity does not update when the **Use real time visualization** checkbox is selected on the Simulation tab of the Run Fluids Analysis dialog.
- When running a Fluids Analysis on Linux, contours do not update when the **Use real time visualization** checkbox is selected on the Simulation tab of the Run Fluids

Analysis dialog. Legends do update as expected.

- When running a Fluids Analysis on Linux, Design Explorer simulations will fail if the **Use real time visualization** checkbox is selected on the Simulation tab of the Run Fluids Analysis dialog.
- When running a Fluids Analysis on Linux, contours do not update and the simulation can crash when the **Use real time visualization** checkbox is selected on the Simulation tab of the Run Fluids Analysis dialog and the model includes rotating components.
- When exporting H3D files on Linux, the application produces an error attempting to start hvtrans.exe.
- As of October 14, 2025, Microsoft Windows 10 will reach its end of support. Following Microsoft's messaging, all Altair 2026 applications will no longer support Windows 10. Altair is providing this information to help our customers prepare and accommodate for this change. Altair 2026 will support the Windows 11 operating system, along with our other Linux-based operating systems. Please contact your local Altair support teams if you have any questions or concerns.