

 RELEASE NOTES

Altair Inspire™ 2021.1 and 2021.1.1

New Features and Enhancements 2021.1

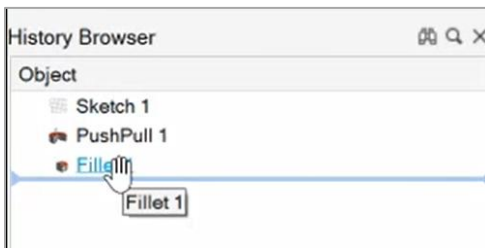
Release Highlights

The Inspire 2021.1 release features the introduction of parametric modeling, as well as new and improved tools for sketching, geometry, and PolyNURBS.

Parametric Modeling

Parametric modeling captures the evolution of your model's shape by showing the linear sequence in which the geometry was created. You can roll the model backward or forward, as well as group, delete, insert, suppress, or edit entities in the History Browser. When you make a change, it's propagated through the sequence of operations and the model is updated automatically.

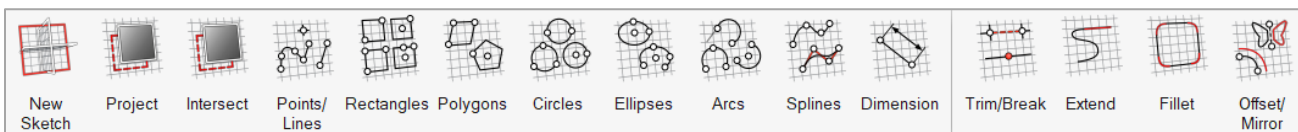
The entities in the History Browser are called "construction features" and they are interdependent. For example, *PushPull 1* in the image below depends on *Sketch 1*, and *Fillet 1* depends on *PushPull 1*. The blue line is the marker, which tells you where you are in the construction history timeline.

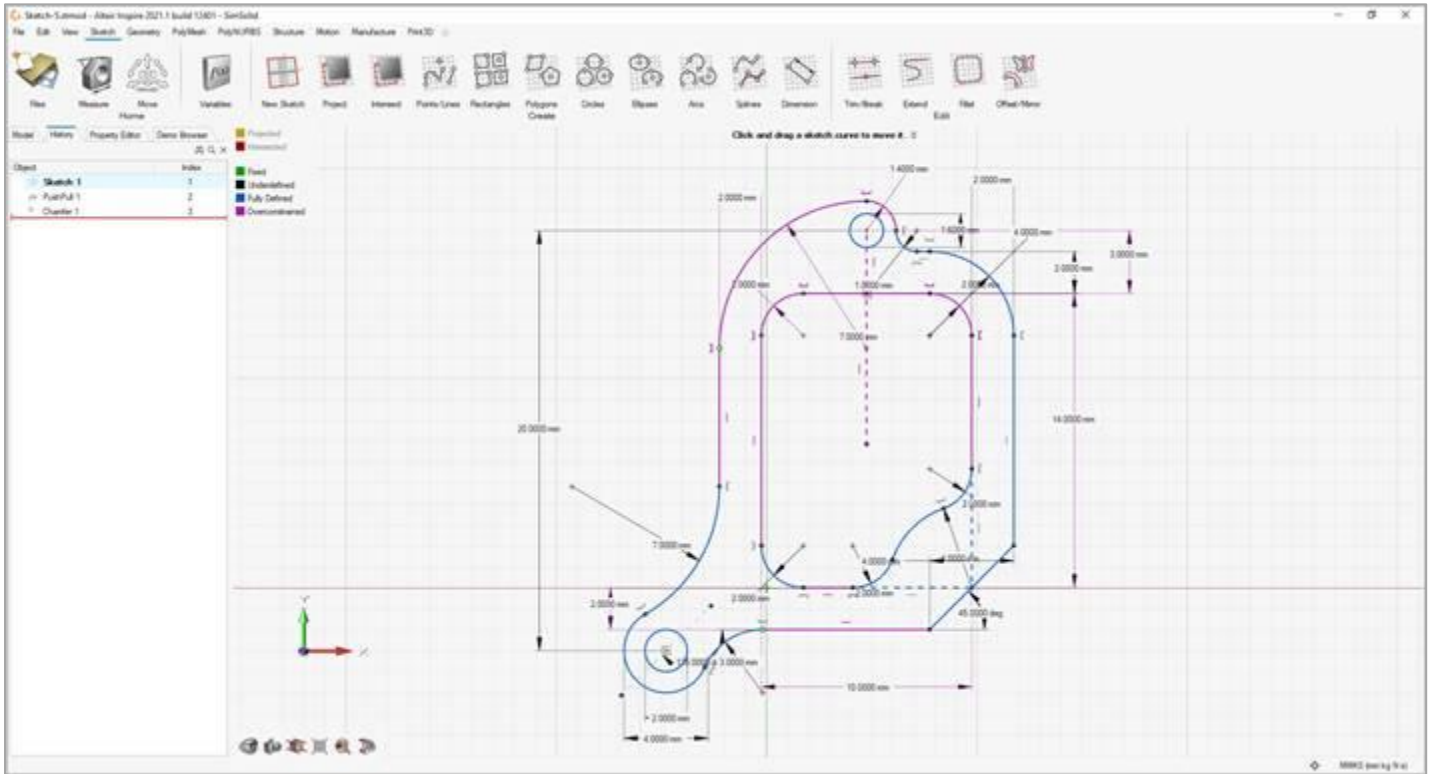


New and Improved 2D Sketching

Inspire 2021.1 features a new and improved sketching interface and a number of new sketching tools. You can now add dimensions, convert driving dimensions to reference dimensions, and add constraints. Highlights include:

- **New sketch tools:** New tools include Splines, Dimension, Extend, Fillet, Offset, Mirror, Project, and Intersect.
- **Improved sketch tools:** All existing tools have been rewritten and improved.
- **Constraints:** A complete set of sketching constraints are now provided including fixed, vertical, horizontal, midpoint, coincident, colinear, tangent, perpendicular, parallel, concentric, and equal.
- **Dimensions, relations, and variables:** All geometry can now be dimensioned, and variables or relations can be used in the dimension definition.
- **Sketch inferencing:** A new sketch inference and snapping engine allows an intuitive method to layout geometry with respect to existing sketch entities.





New Geometry Modeling Tools

New tools have been added to the Geometry ribbon including Reference Geometry, Extract, and Shell.



PolyNURBS and Facet Improvements

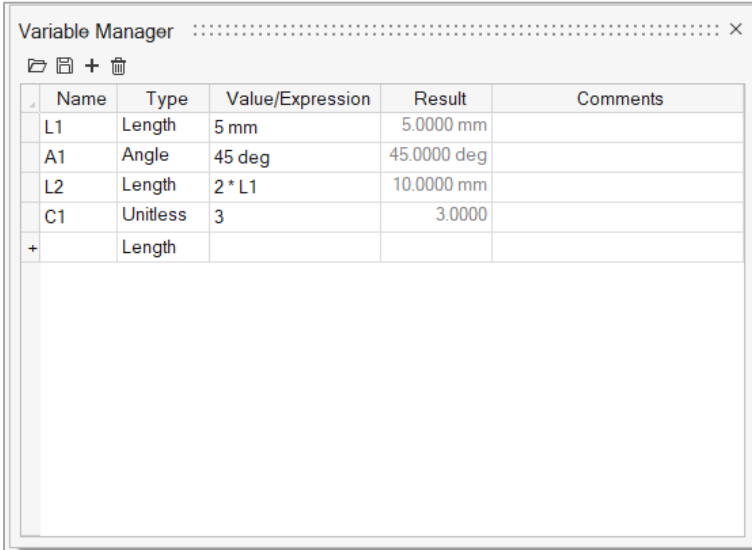
Two new PolyNURBS tools have been added to Move Bodies and Mirror Bodies. PolyNURBS now retain all downstream geometry operations such as Booleans and fillets even after manipulating the cage.



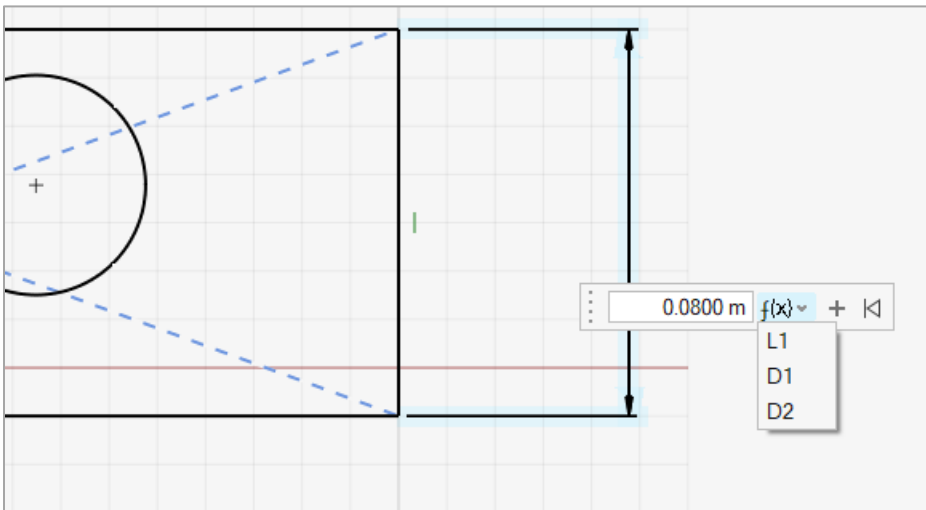
Variable Manager

Geometry based variables can be defined and referenced through the application providing unique flexibility to configure and explore design variations.

Use the Variable Manager to create new variables, edit variables, and import or export variables to a .csv file. You can then add and define variables for use with the sketching and geometry tools. You can also specify dependencies between the variables, which will cascade changes and update the model accordingly.

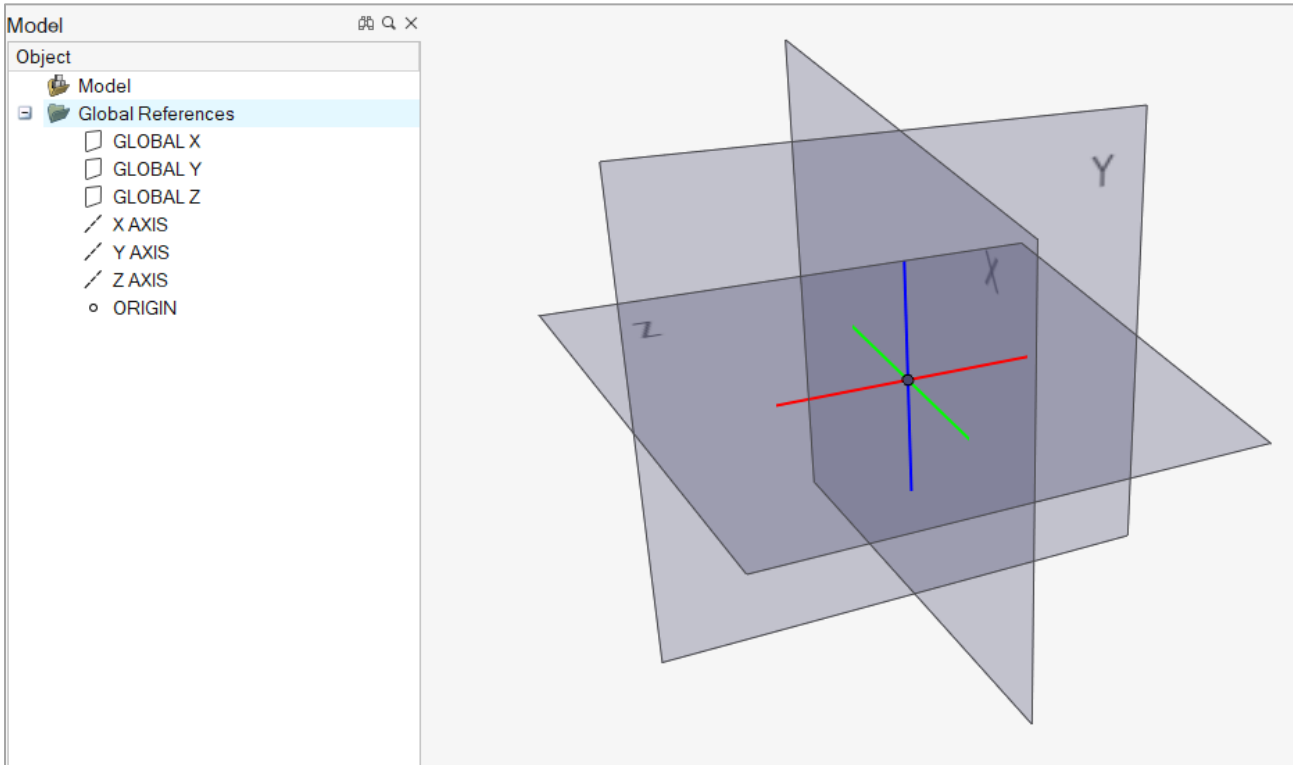


Once defined, you can then apply variables when using the sketching and geometry tools from the tool microdialogs. You can select any available variable previously defined in the Variable Manager whose type matches the selected field. Click the f(x) button to choose from the available variables. You can also dynamically create a variable using the Add Variable button and typing the variable.



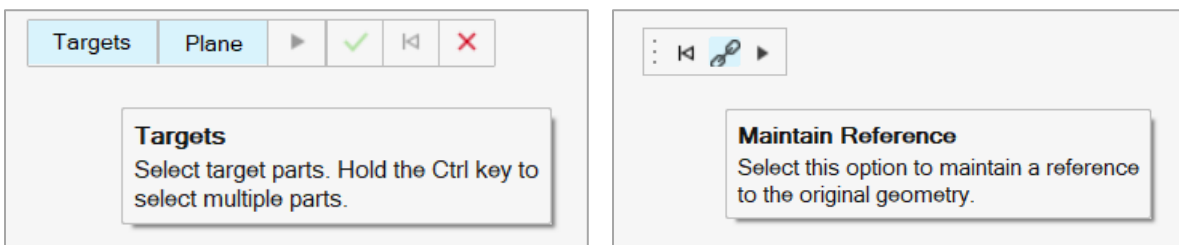
Global References

Inspire now includes global references for the origin, global X, Y, and Z planes, and X, Y, and Z axes. These are hidden by default, but can be turned on by clicking the corresponding icon in the Model Browser.



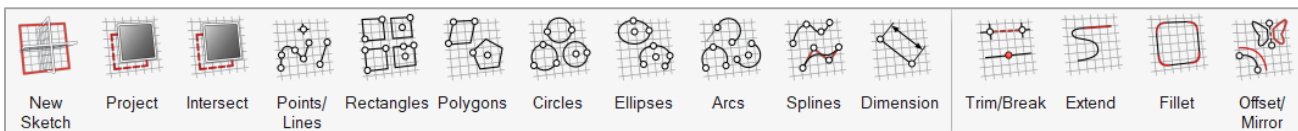
Enhanced Tooltips

Enhanced tooltips have been added to microdialogs and guide bars for sketch, geometry, and PolyNURBS tools. This allows you to learn more about the tool parameters and options without needing to open the help.



Sketching

The new parametric sketching ribbon now features a constraints legend and variety of new tools. You can apply variables when using most tools, and edit sketches using the new History Browser (F6).



Constraints Legend

Inspire now provides feedback as to whether a sketch is under-constrained or over-constrained in the new sketch legend.



Project

Project selected geometry onto the sketch plane.

Intersect

Extract curves resulting from the intersection of the sketch plane with selected parts.

Polygons

Sketch a regular polygon by defining the center and radius, or sketch a parallelogram by defining three corners.

Ellipses

Sketch an ellipse by defining the center, width, and height. You can also create an open ellipse by adding start and end points.

Splines

Sketch a spline curve using fit points or controls points. When using the Spline Through Points tool, the spline curve will pass through the sketch points. Splines Using Control Points will pass near, but not through, the sketch points.

Dimensions

Apply and edit a dimensional constraint to control the size and proportions of a sketch entity.

Extend

Extend or shorten a sketch entity.

Fillet

Round the corners of a sketch entity to create fillets.

Mirror

Mirror selected sketch entities about an axis.

Offset

Offset selected sketch entities by clicking and dragging the original sketch.

Additional Changes and Enhancements for Sketching

- In the Polyline tool, you can now use the Shift key to toggle between creating a line and an arc.
- The Create Imprint on Face option is now available in the right-click context menu for sketching
- A new sketching section has been added the Preferences. This includes an option to show or hide intersection curves on sketch plane creation.
- Sketch entities cannot be moved directly with the Move tool, as they are tied to the sketch plane. To move a sketch, you must either move the sketch plane or transfer the sketch to a different sketch plane

Geometry

The updated parametric Geometry ribbon now features new tools for Reference Geometry, Extract, and Shell. You can apply variables when using most tools, and edit sketches using the new History Browser (F6).



Reference Geometry

Create reference planes, references axes, and reference points from geometry features or other reference entities.

Extract

Extract selected geometry features and transfer them to a new part.

Shell

Remove material and create thin walls to generate a shelled part.

Additional Changes and Enhancements for Geometry

- The Move tool has been updated so that moves can now be referenced.
- The Push/Pull tool now allows you to add, subtract, replace, or create a new part from the result of sketch face that has been pushed or pulled. The default is to add to the current part.
- The Keep Original and Instance options for the Mirror tool have been moved to the Find Options menu.
- The visualization for Boolean tools has been updated.
- The Cut tool has been renamed Slice.
- The Midsurface tool now remembers your previous selection for Mid vs. S1/S2.
- You can now purge parts and assemblies by selecting Delete Without History from the part context menu or the Model Browser context menu. A Remove History option is also available on the right-click context menu for the Delete Parts construction feature in the History Browser timeline. Selecting this option will purge the construction feature from the timeline.

PolyNURBS

The updated parametric PolyNURBS ribbon now features new tools to move and mirror PolyNURBS bodies. PolyNURBS now retain all downstream geometry operations such as Booleans and fillets even after manipulating the cage.

You can edit PolyNURBS using the new History Browser (F6).



Move Bodies

Translate or rotate PolyNURBS bodies.

Mirror Bodies

Mirror the bodies of a PolyNURBS part around a symmetry plane.

Subdivision Surface

Select the Subdivision Surface check box in the upper right corner of the modeling window to maintain the part as a mesh-based subdivision surface rather than converting to NURBS surfaces. This makes it much faster to exit the tool, as it won't perform the NURBS surface conversion.

Additional Changes and Enhancements for PolyNURBS

The following changes and enhancements have also been added for version 2021.1:

- PolyNURBS parts can now be edited by double-clicking the part or by editing the Construction Feature in History Browser.
- The Close tool has been renamed Repair.

Structures

Updates to Contact Gap and Penetration Values for SimSolid Analysis

The default contact settings are now the same for both the SimSolid and OptiStruct solvers.

Reset Refined Results for SimSolid Analysis

You can now reset any SimSolid results you've refined in the Analysis Explorer.

Additional Changes and Enhancements for Structures

The following changes and enhancements have also been added for version 2021.1:

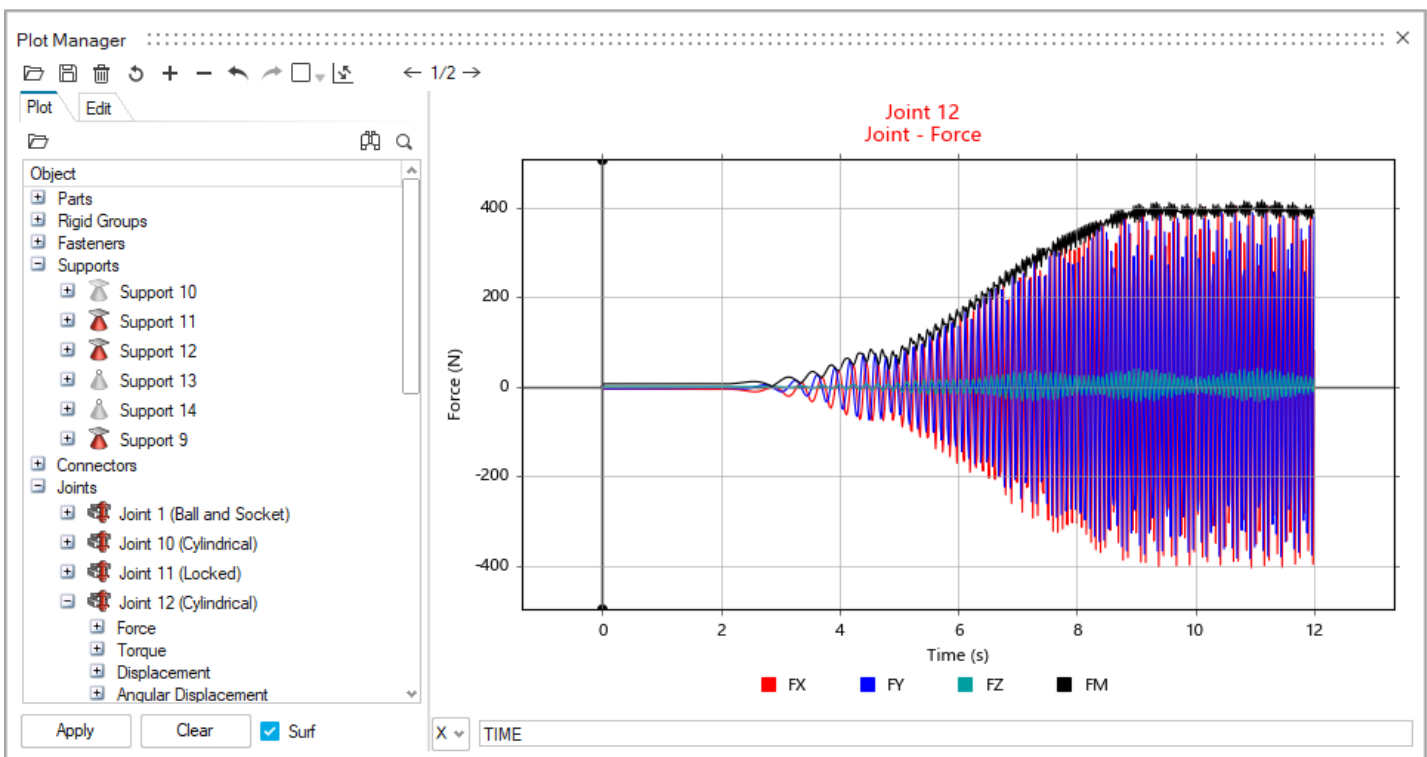
- Section cuts can now be visualized in part color using the Preference setting under Inspire > Visualization.

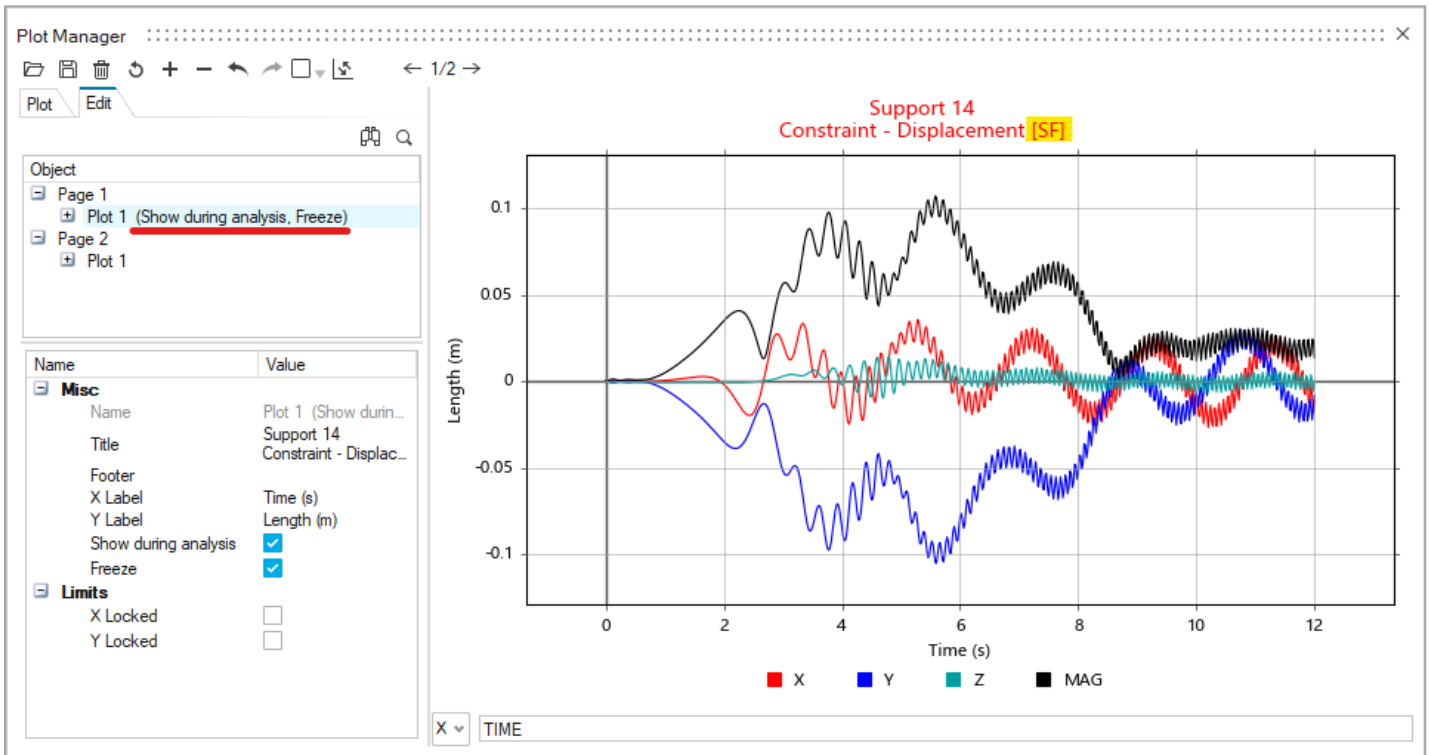
Motion

Plot Manager Improvements

Improvements have been made to the Plot Manager for motion post-processing, including:

- Showing entity icons in the Plot Browser that can also reflect whether entity is currently hidden or not.
- Renaming of containers such as Supports (instead of Constraints) and Connectors (instead of Spiders) in the Plot Browser.
- Denoting when a plot has had **Show during analysis** or **Freeze** enabled.
- Automatically dismissing plots when exiting the Review Motion Results tool.
- Using an internally stored representation of plot template instead of using disk-based autosave operations.
- Improved template robustness for handling cases when some model entities have been suppressed.





Parts List and Report File for Rigid Groups

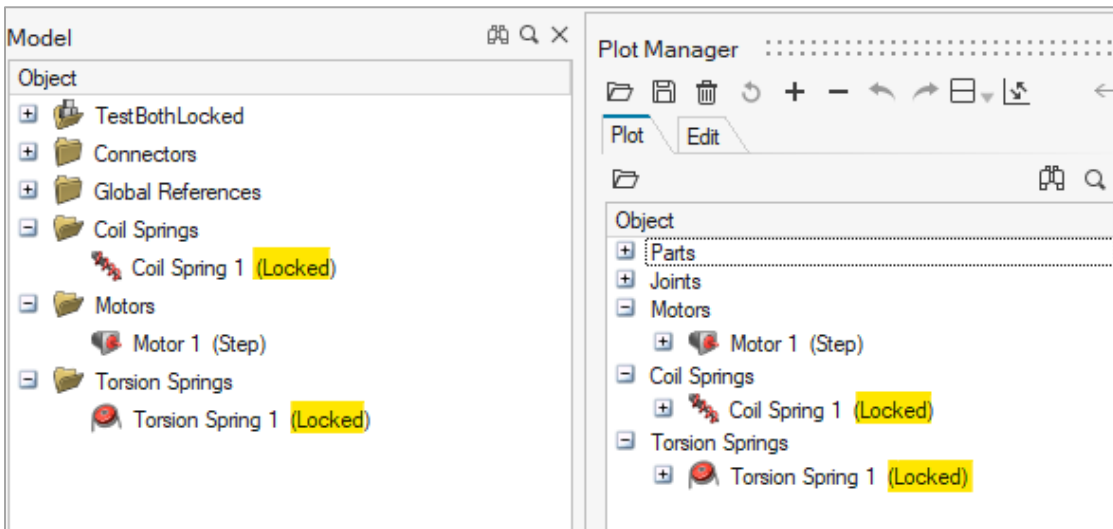
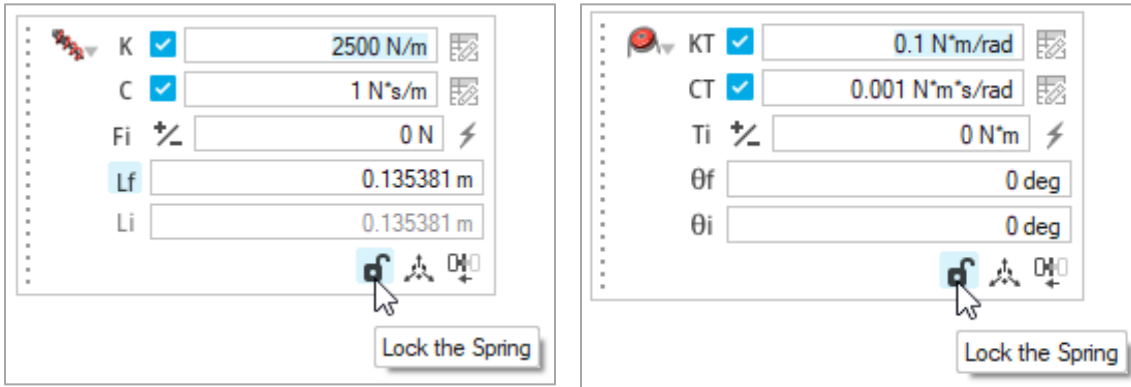
The Rigid Groups table now includes a Parts column that lists the parts contained in each rigid group. Additionally, there are now buttons to save and view a rigid groups report file. This report contains more detailed information including the mass, mass moment of inertia, center of gravity location, and more.

Name	Parts
Rigid Group 1	Geneva Wheel, Geneva ...
Rigid Group 2	Driver, Cam

Lock Option for Spring Dampers

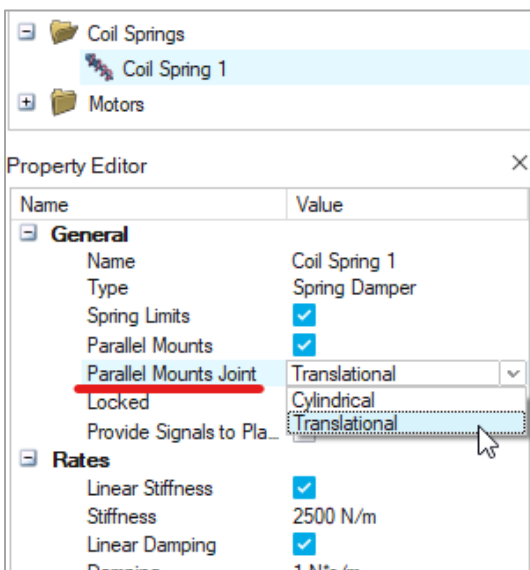
The Coil Spring and Torsion Spring microdialogs now have a Lock button similar to the lock feature for motors and actuators. When locked, the spring-dampers do not deform, and you can plot the Lock Force (for coil spring) and Lock Torque (for torsion spring) to understand the loading or to determine the preload required for the design position.

For convenience, you can also find the Lock checkbox available on the context menu for springs, as a column in the springs table, as well as in the Property Editor under the General category. The locked state is also reported in the Model Browser and the Plot Browser in Plot Manager.



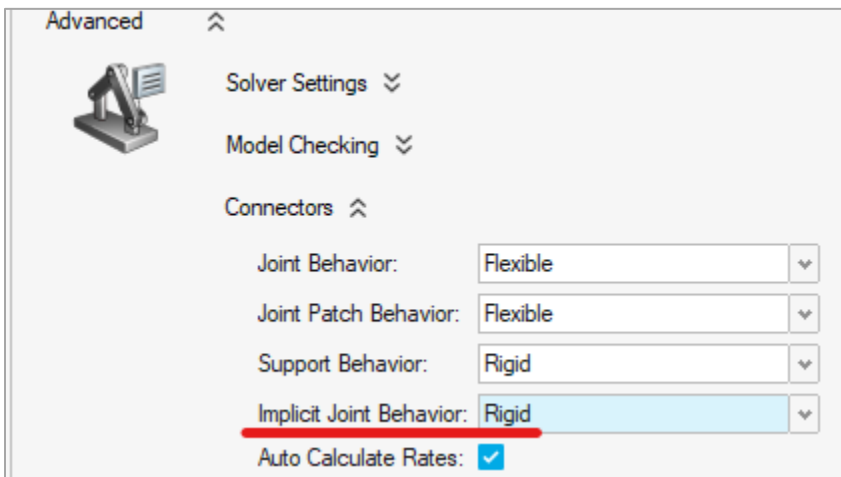
Implicit Joint for Coil Springs

The Parallel Mounts Joint property for coil springs now offers a Translational implicit joint as an alternative to the previously available Cylindrical joint.



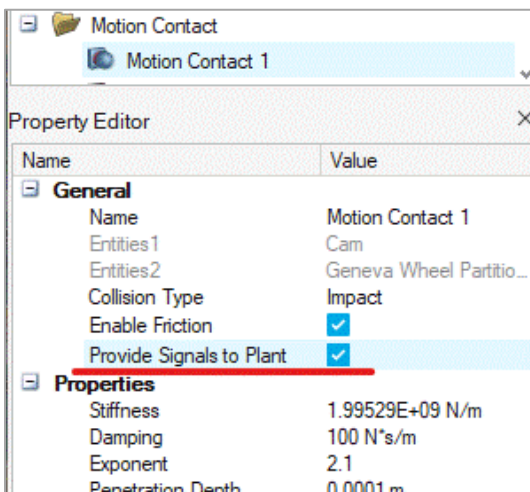
Implicit Joint Behavior Setting

An option for specifying Implicit Joint Behavior has been added to the Run Settings dialog in the Connectors category under the Advanced section. Rigid is the default setting for implicit joint behavior.



Export MDL with Plant Output Signals for Motion Contacts

The Provide Signals to Plant option is now available for motion contacts, but is disabled by default. It can be found in the Property Editor and when enabled, the motion export operation will include contact force output plant signals in the .mdl file for use in MotionView, MotionSolve, and Altair Activate.



Additional Changes and Enhancements for Motion

The following changes and enhancements have also been added to Inspire Motion for version 2021.1:

- Expanded range for damping in coil spring-dampers:** We now allow entry of 0.0 for the coil spring damping rate. This is useful when you want to study the effect of having no damping without altering the spring-damper type.

- **New filter buttons on Force Explorer:** More buttons have been added to the Force Explorer for connectors and fasteners.
- **Animation (H3D) file writing improvement:** If enabled in the Motion Run Settings, this file will now be created even if you stop the run manually or if the run fails during the motion analysis.
- **Jump to Contact Event improvement:** The Jump to Contact Event buttons on the Force Explorer now work within the start and end times of the animation range specification.

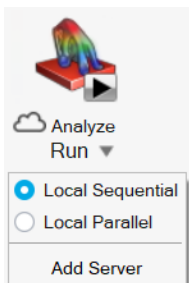
Print3D

Improved Geometry Integration

Parametric geometry that is created in Inspire is now seamlessly integrated with Print3D. Edits made to geometry using construction history are carried over and automatically reflected in Print3D operations.

Run Jobs on a Remote Server

You can now run jobs remotely on an Altair PBS server. You can also choose to locally run Print 3D jobs either sequentially or in parallel.



Resolved and Known Issues

Resolved and known issues are listed in the online help. To view the latest information, go to **File > Help > What's New**.

New Features and Enhancements 2021.1.1

Altair Inspire 2021.1.1 includes the following new features, enhancements, or resolved issues.

Resolved Issues

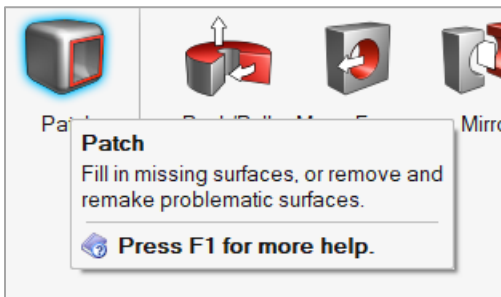
- Fixed an issue where you could not perform a quick analysis run (F8) with the Variable Manager open. [IN-25524]
- Target mass calculation has been corrected for gauge optimization. [IN-25659]
- SimSolid Error 17, numerical instability, has been fixed. [IN-25480]
- Fixed an issue with construction history features failing to recompute after saving or opening a file. [IN-25460]
- Editing an assembly that is mirrored no longer causes a crash. [IN-25503]
- Fixed an issue removing intersected parts from inside the sketch when they are left behind because of upstream changes. [IN-25467]
- Fixed an issue where PolyNURBS exported in OBJ format were being written as facets instead of cage faces. [IN-25559]
- Fixed an issue editing Move Face construction features. [IN-25604]
- Corrected several translation issues found within the product. [IN-25491, IN-25492, IN-25551, IN-25626, IN-25651, IN-25461, IN-25459, IN-25458]

Learn More About Inspire

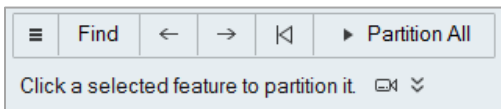
You can learn more about new and existing features in Inspire using the following resources:

In-Application User Assistance

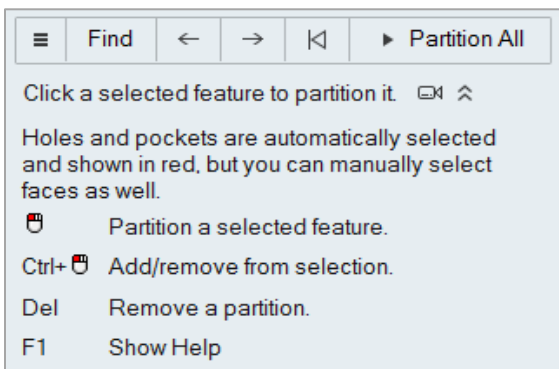
Inspire provides two types of user assistance. **Enhanced tooltips** appear when you hover over icons and other features. They describe what the tool does.



Workflow help appears when you select a tool that opens a guide bar or microdialog. The text prompts you what to do next.

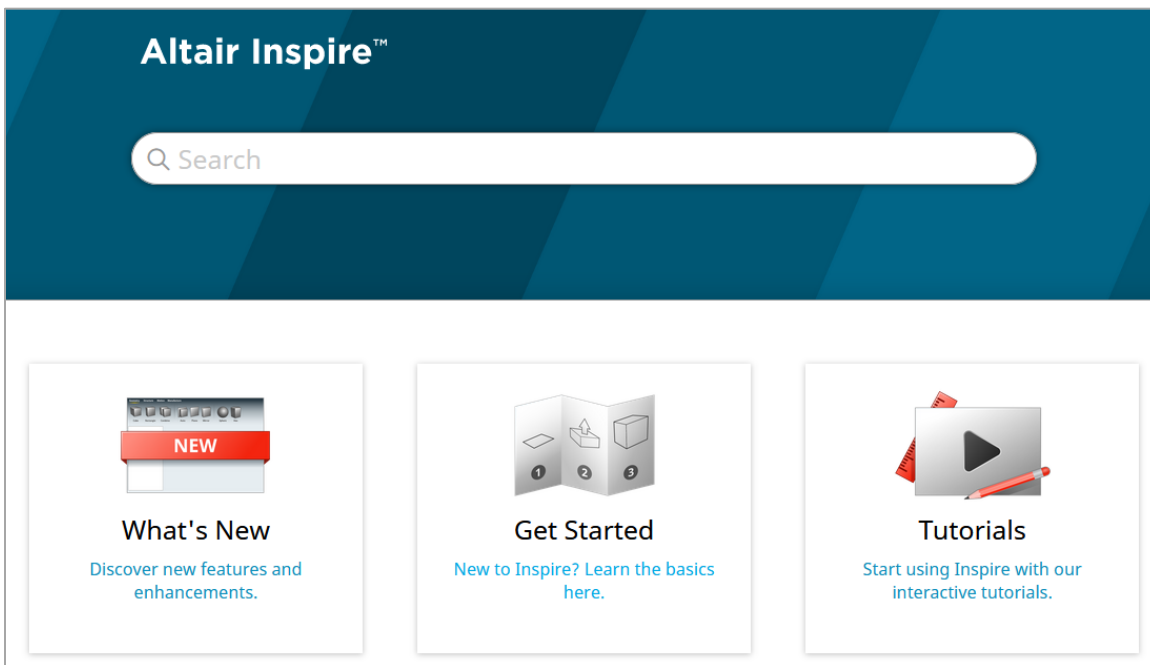


Click to view additional tips and shortcuts. Some tools also include a video .



Online and Offline Help

Press **F1** or select **File > Help > Help** to view the online help.



You can download an offline version by selecting **File > Help > Download Offline Help**. An internet connection is required to download.

