Altair Inspire Cast 2021.1

# New Features and Enhancements 2021.1

Release Highlights

The Inspire Cast 2021.1 release features new and improved tools for sketching, geometry, and PolyNURBS. It also includes the addition of mesh component factors and an improved shock capturing solver.

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 exiting 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.
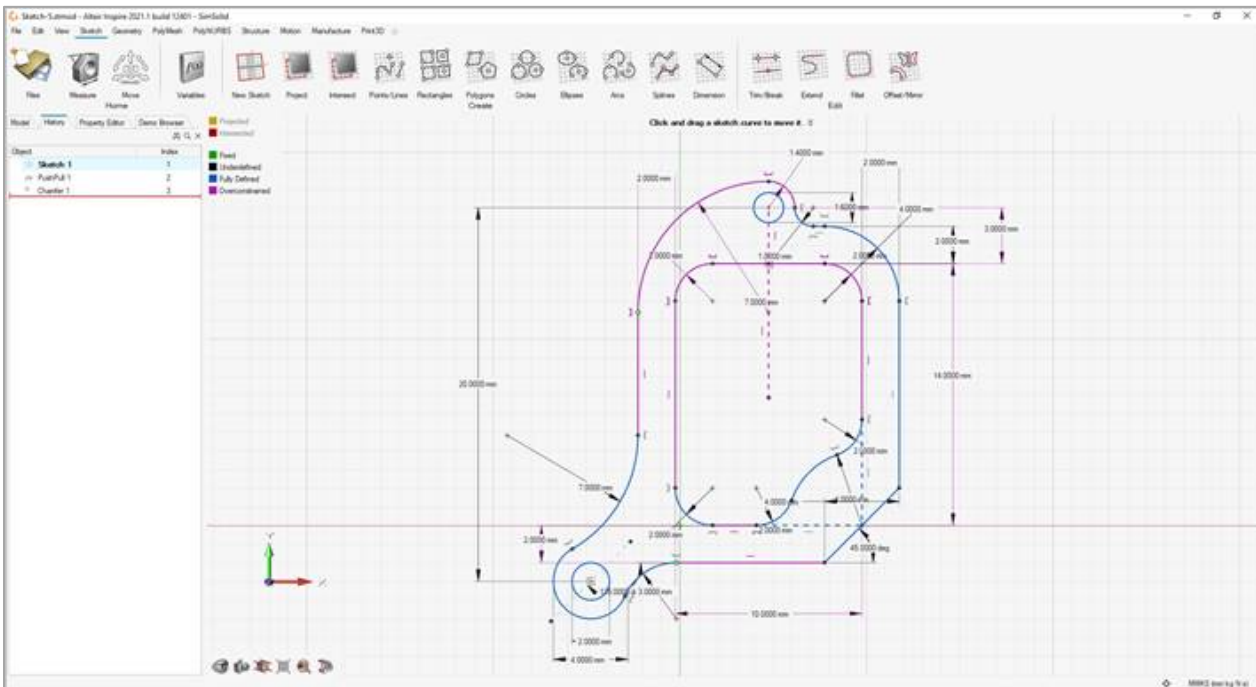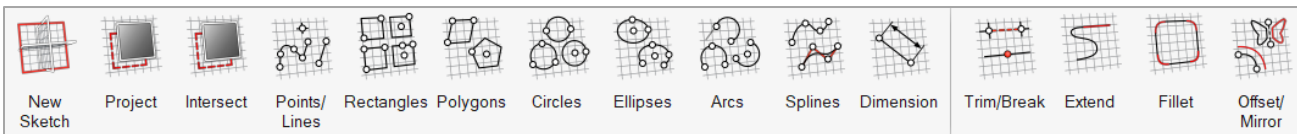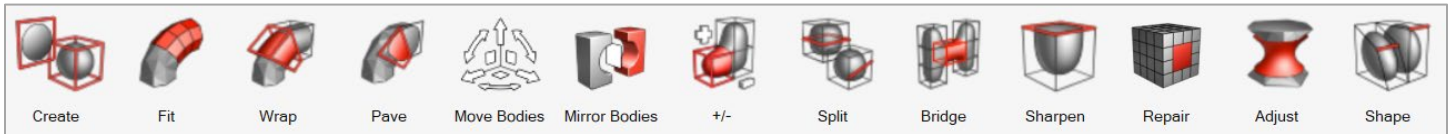
New Geometry Modeling Tools

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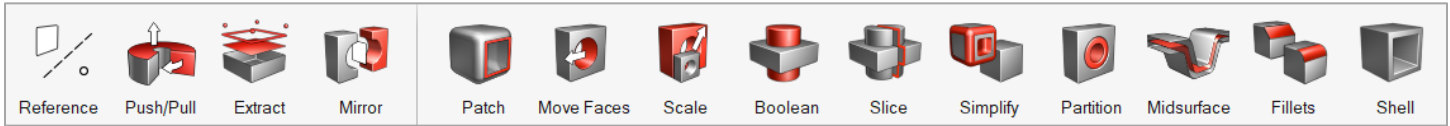
PolyNURBS and Facet Improvements

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.

Global References

Inspire Cast now includes global references for the origin, global X, Y, and X 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.
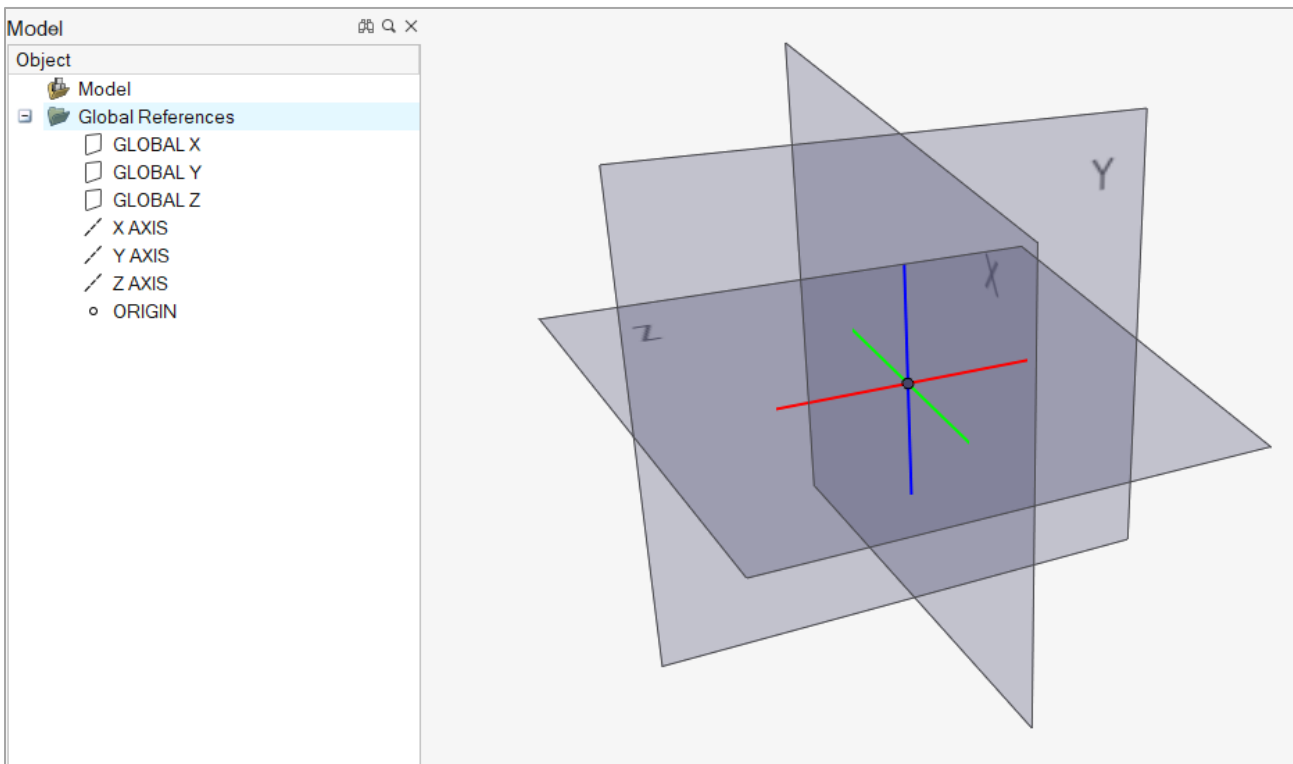
Enhanced Tooltips

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.
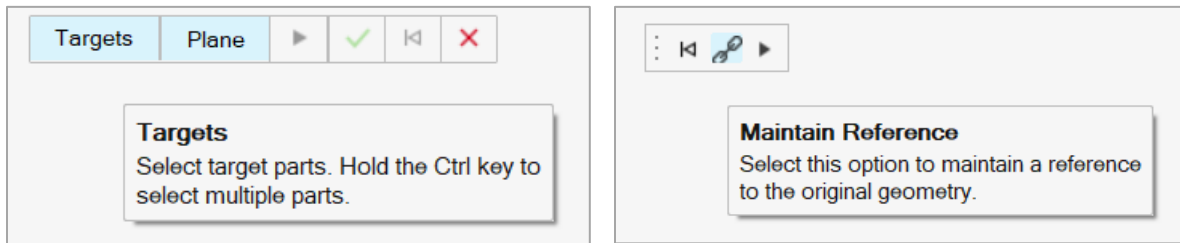
Sketching

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).
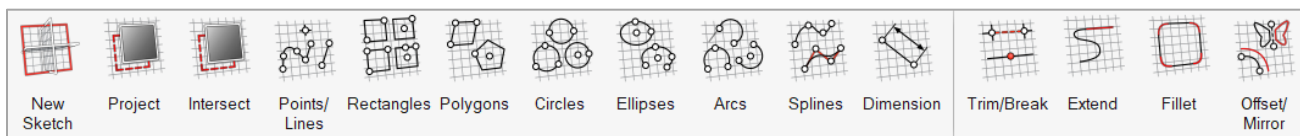
Constraints Legend

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.
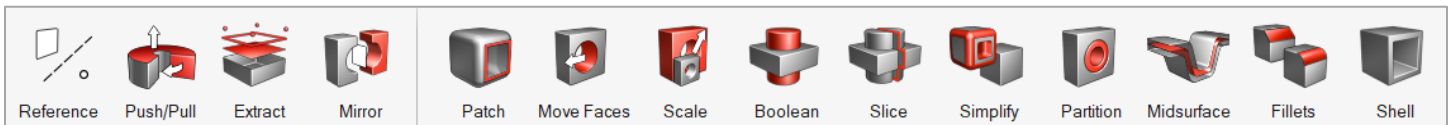
Additional Changes and Enhancements for Sketching

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

- In the Polyline tool, you can now use the Shift key to toggle between creating a line and an arc.
- 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.
- 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.
- Move tool selection has been disabled on realized sketches.

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

- The Move tool has been updated so that moves can now be referenced.

- The Push/Pull tool now allows you to combine, subtract, or replace the result of sketch face that has been pushed or pulled.

- 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.

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:

- The Edit PolyNURBS tool has been removed, as editing is now handled through the History Browser.

- The Close tool has been renamed Repair.

Casting

The Casting module now offers additional component mesh factors, as well as a new filling solver.

Component Mesh Factors

Component mesh factors are now available on the Advanced tab in the Run Analysis window for casting. These options can be used to automatically improve the mesh in places where this is needed, such as small thickness sections, or to force a minimum number of layers. Options include:

- **Liquid grade factor**: Edit the transition (how the mesh grows) between two different bodies (part, runner, overflow, riser. etc.) that are in a liquid state during casting.

- **Solid grade factor**: Edit the transition (how the mesh grows) between two different bodies (chiller, mold, sleeves, etc.) that are in a solid state during casting.

- **Auto refinement**:
    - **Minimum thickness**: If the distance between surfaces are within the mínimum thickness, the mesh will be refined on these surfaces.
    - **Minimum size**: The mesh refinement will not create elements with edges smaller than the specified minimum size.
    - **Minimum number of layers**: The mesh refinement creates layers of elements in thin sections inside a body. The number of layers can be more than **minimum number of layers** defined here provided that the edge length is greater than the **minimum size**.

Improved Shock Capturing Solver

The casting solver has implemented a new fluid solver that includes a formulation to impose weak slip conditions. This provides more accurate results in thin-to-thick transition regions (such as ingates). The solver also includes a new shock capturing term that improves convergence.

The new solver can be enabled in the Preferences under Inspire Cast > Analysis.

## Casting Modulus Using Geometrical Analysis

A new method for computing the casting modulus based on geometric and solidification data is available in the Analysis Explorer under Solidification Results. Data image processing is used to post-process the solidification pattern and then divide the part to compute this modulus.

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.
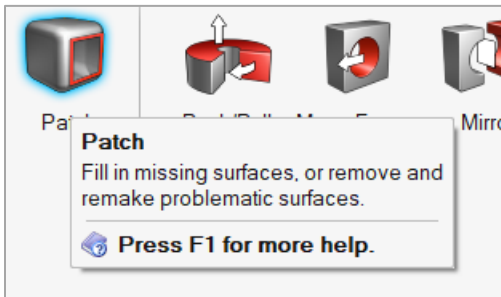
Workflow help appears when you select a tool that opens a guide bar or microdialog. The text prompts you what to do next.
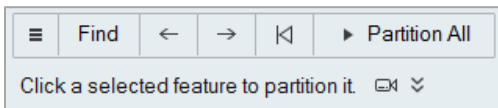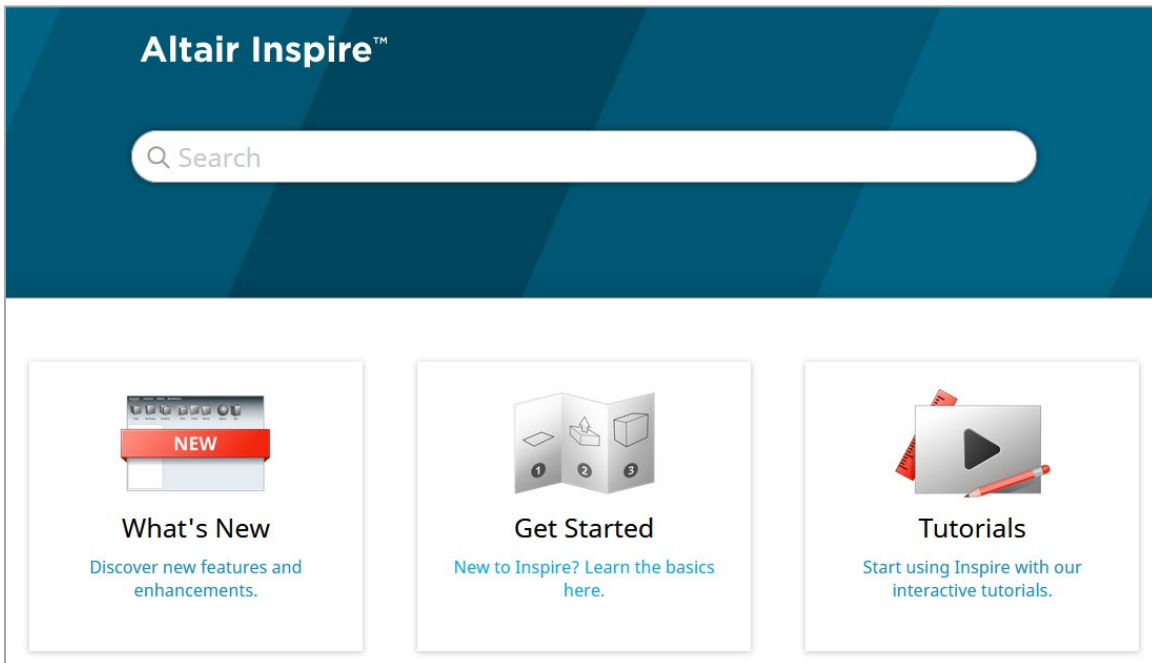
Click ⌄ to view additional tips and shortcuts. Some tools also include a video ▭◁ .

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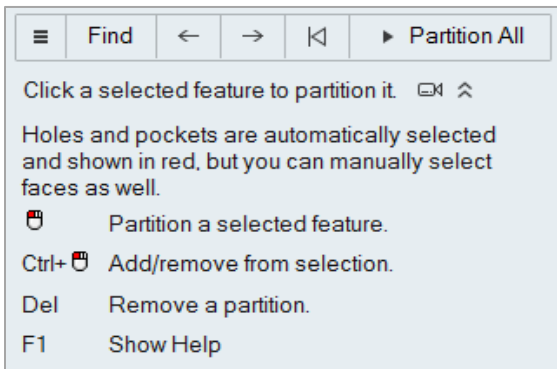
Online and Offline Help

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.