

 RELEASE NOTES

Altair Inspire Form 2021.1

New Features and Enhancements 2021.1

The Inspire Form 2021.1 release includes the following new features and enhancements.

Feasibility

New additions and improvements to the forming feasibility module include:

Unflange / Flange Open

Bends and flanges along a linear path can now be opened and developed using the Flange tool.

Cutout

The Cutout tool has been improved to automatically recognize whether selected edges are concave or convex and fill them in accordingly.

Orient

Draw direction orientation can now be defined by selecting a system and/or surface not associated with the part.

Blank shape

A new option has been added in the Preferences to export blank shapes as output from solver without edge smoothing after a feasibility run. By default, this Preferences is set to Off.

Post Processing

Results can be now visualized as a thickened part for better interpretation and understanding.

Tryout

New additions and improvements to the forming tryout module include:

Draw Direction and Cam Tool Direction

Tool motion direction can be defined by selecting a user defined system. By default, Z axis of the selected system is set as draw direction.

Gravity

Gravity run performance and accuracy has improved due to a new solver. In addition, gravity runs can now be made using a coarse mesh setting to significantly reduce computation time.

Note: Only the deformed shape, due to its weight, is available as results. Animation and other results including displacement are not available.

Post processing

Several improvements have been made to post-processing, including:

- Performance improvements when loading and animating results from large models.
- A new option to animate all of the operations results in a sequence.
- New options in the animation play bar to quickly navigate between operations.

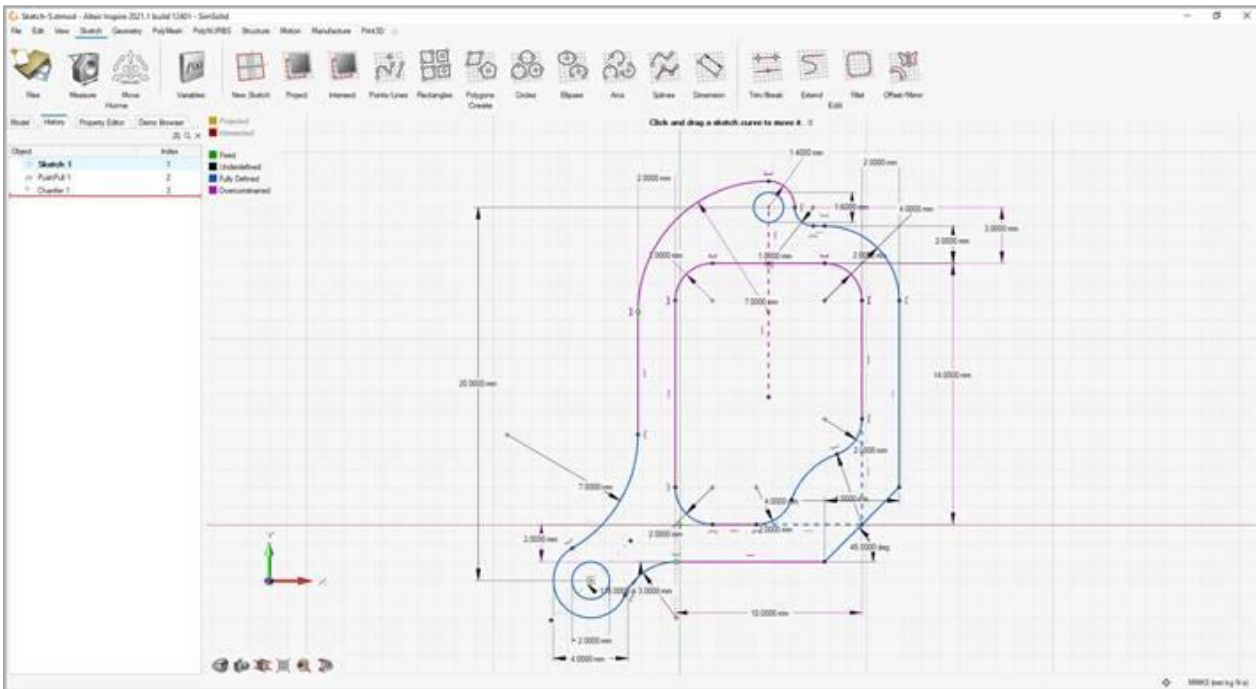
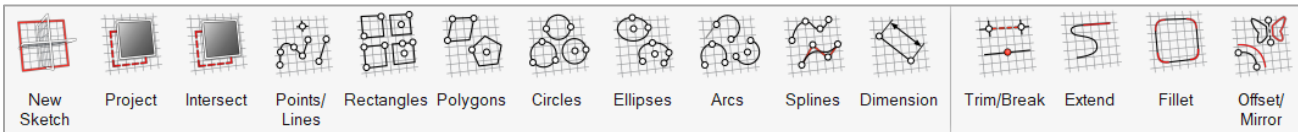
Tryout Run Options

The number of processors for local runs can be now defined in the Preferences. By default, it is set to Auto.

Sketching

The new parametric sketching ribbon now features a constraints legend and variety of new 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 inferecing:** A new sketch inference and snapping engine allows an intuitive method to layout geometry with respect to existing sketch entities.



Constraints Legend

Inspire Form 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.

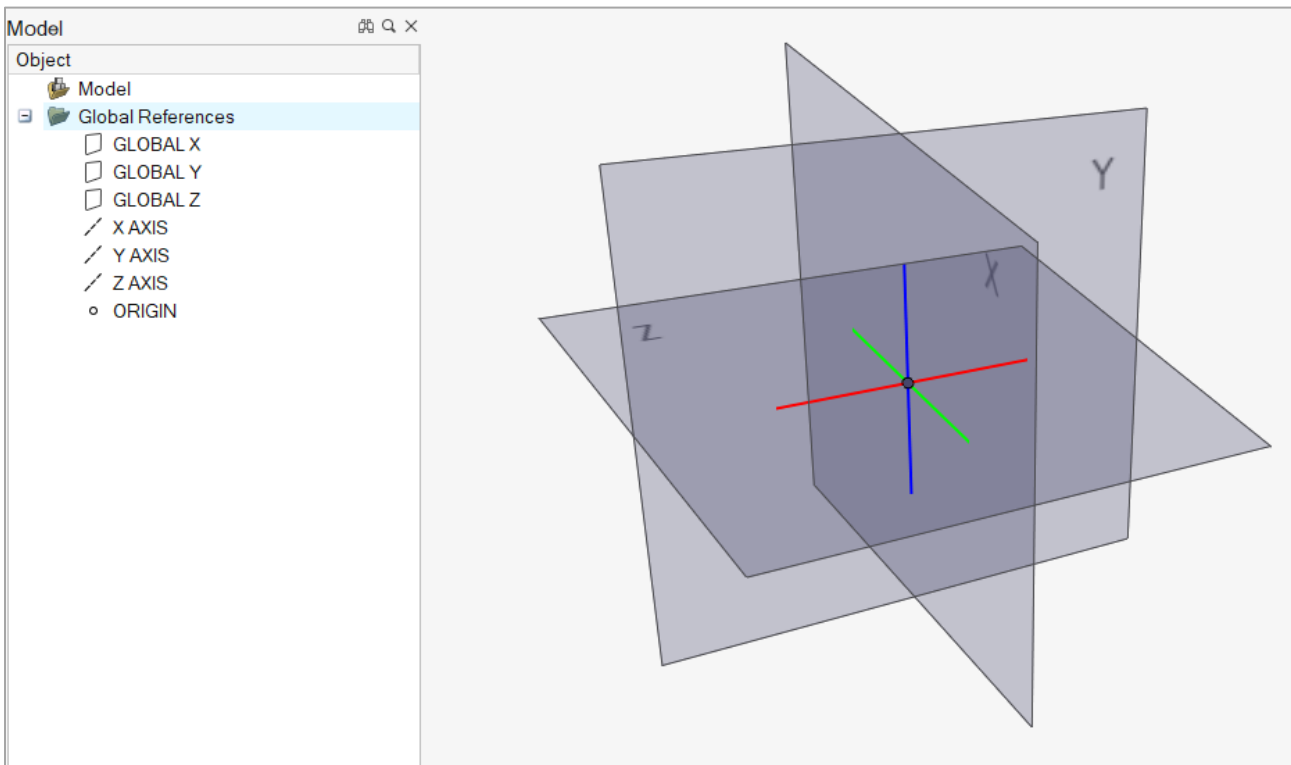
Geometry

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).



Global References

Inspire Form 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.



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.

Resolved Issues

The following issues have been resolved for the 2021.1 release:

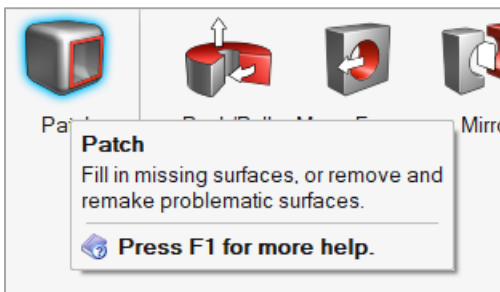
- Callout font size in published reports have been increased for ease of reading.
- Fixed an issue with callouts being truncated in the published report.
- Fixed several issues in trimming operations related to missing surfaces/elements, deleting the wrong surface/elements, and failure to trim.
- Fixed an issue where the edge-to-carrier distance was applied twice for the center carrier.
- Fixed an issue where it was not possible to restart after a failed run.
- Fixed an issue where the tool closed position was reset to manual mode after review.
- Fixed an issue where it was not possible to import multiple FE mesh files.

Learn More About Inspire Form

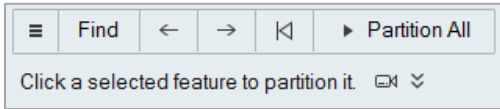
You can learn more about new and existing features in Inspire Form using the following resources:



In-Application User Assistance

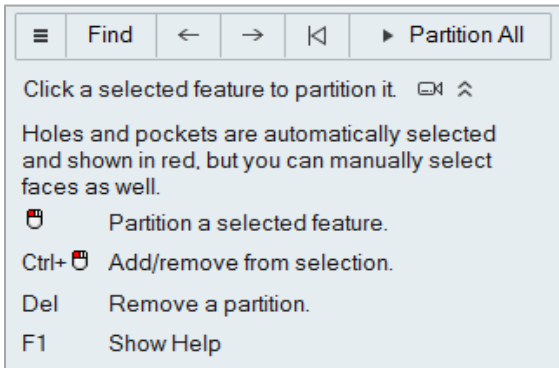
Inspire Form provides two types of user assistance. **Enhanced tooltips** appear when you hover over icons and other features. They describe what the tool does.



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

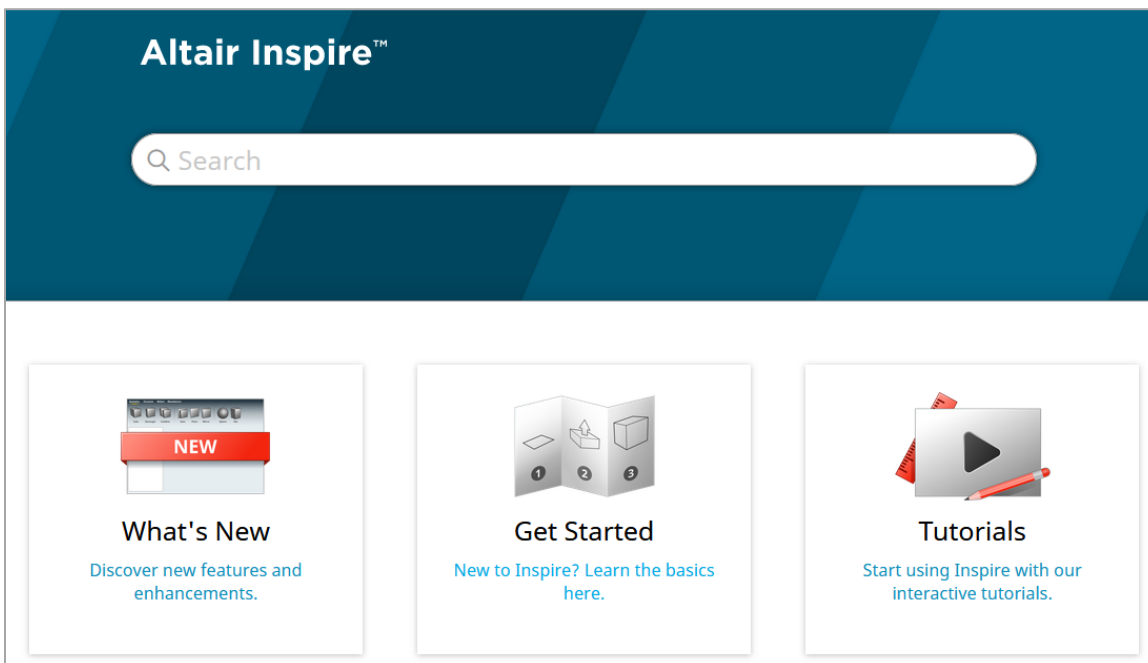


Click  to view additional tips and shortcuts. Some tools also include a video .



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.

