Altair Inspire Extrude

2021.1
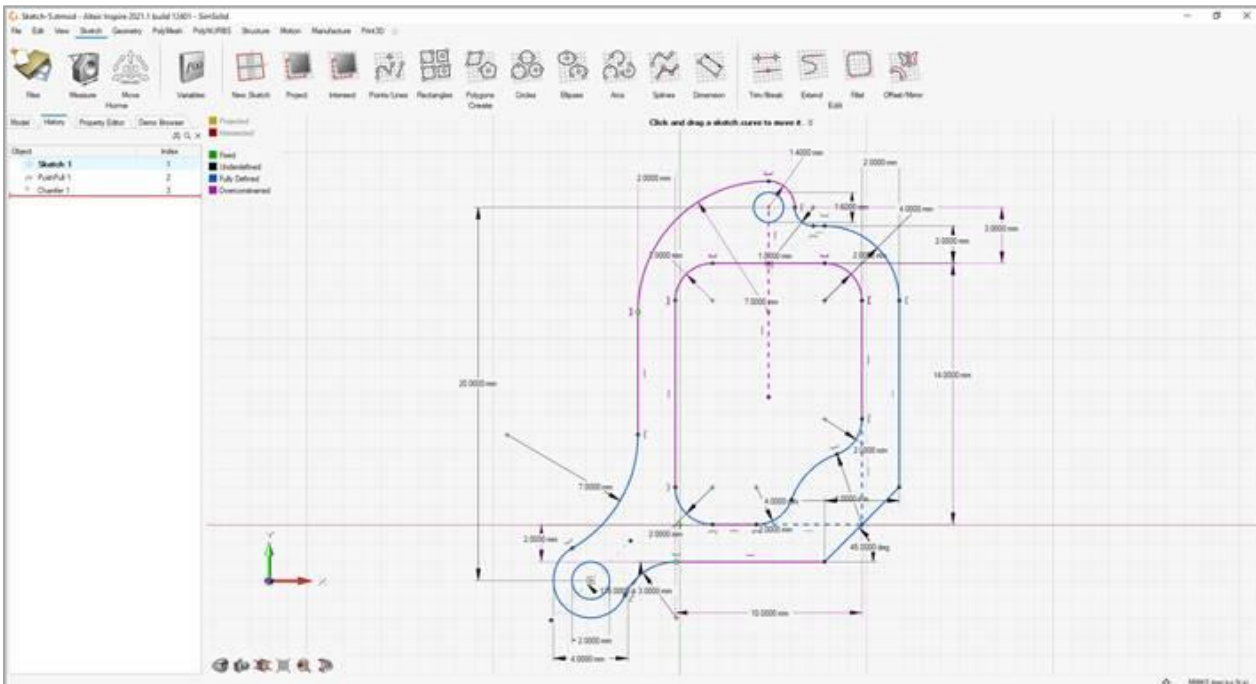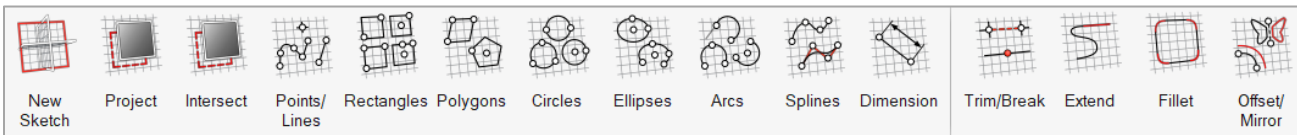
New Features and Enhancements 2021.1

Release Highlights

The Inspire Extrude 2021.1 release features new and improved tools for sketching and geometry.

New and Improved 2D Sketching

Inspire Extrude 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.
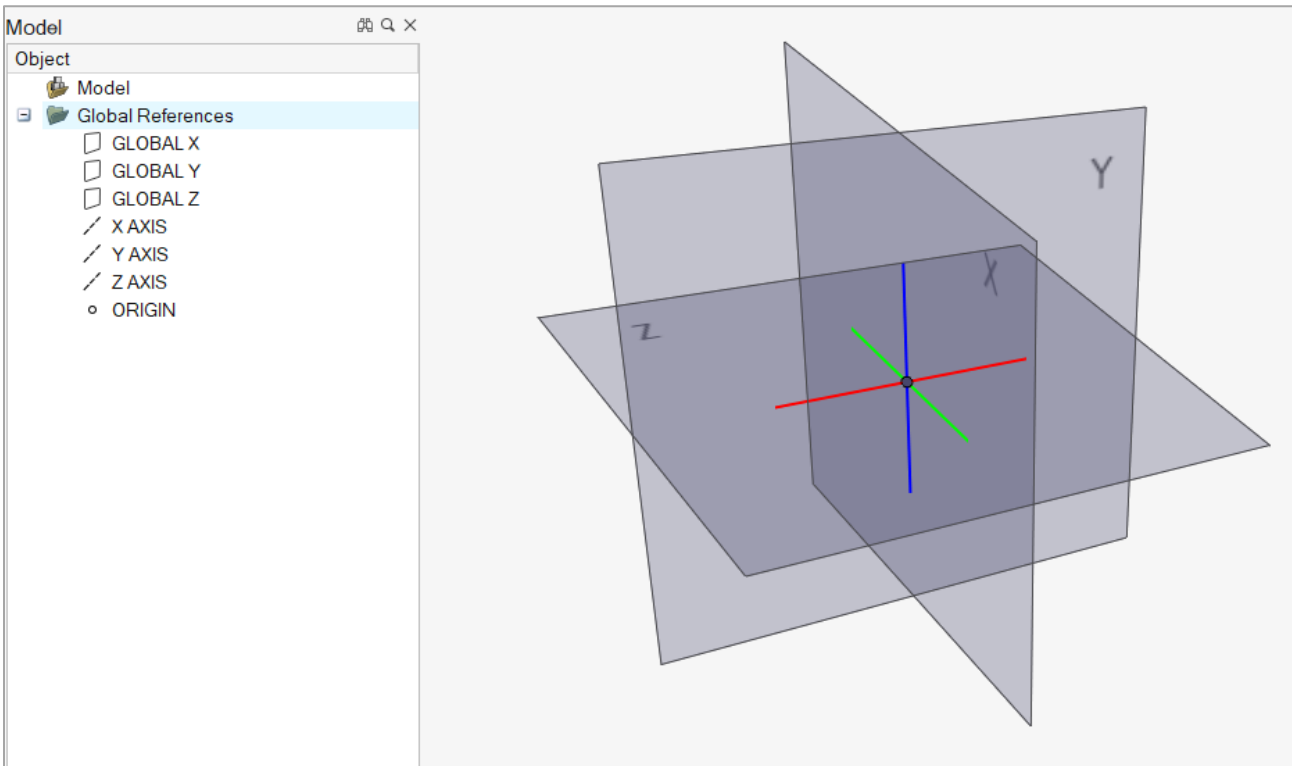
New Geometry Modeling Tools

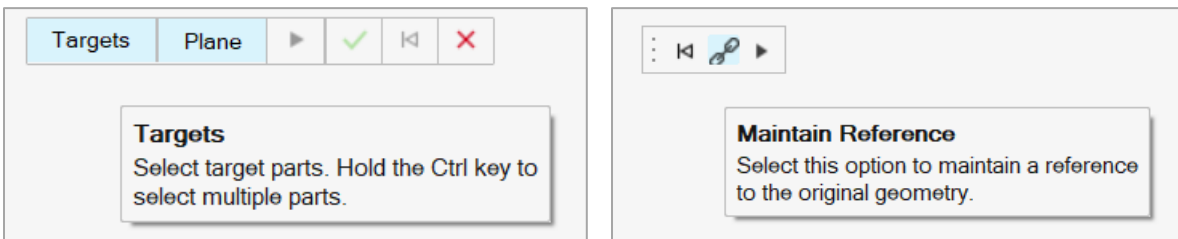New tools have been added to the Geometry ribbon including Reference Geometry, Extract, and Shell.

Global References

Inspire Extrude 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.
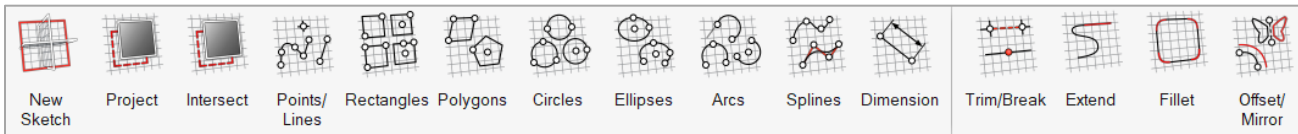

Enhanced Tooltips

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 Extrude 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.
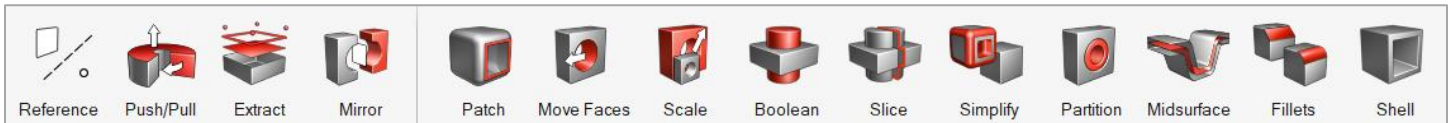
Additional Changes and Enhancements for Sketching

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.

Metal Extrusion

The following changes and enhancements have been added to Inspire Extrude Metal for version 2021.1:

Process Data Improvement

HTC BC was previously applied only at the surface level. In this release, this feature is available at the part level, simply click on the preferred part and enter the HTC data in the dialog. [IEXT-1989]

Improvement in Tool Deflection

The batch mode mesh process for the Tool Deflection model setup has been improved in this release. Issues related to CAD ID duplication and face mesh extraction have been resolved. [IEXT-2165, IEXT-1983]

HyperXtrude Solver Suite

Altair HyperXtrude is a suite of finite element solvers for simulating the following manufacturing processes.

- Metal Extrusion

- Polymer Extrusion

- Quenching

- Calibration

- Metal Rolling

- Friction Stir Welding

- Resin Transfer Molding

New Features

Updates to the H3D Writer Module

H3D files exported from the solver now use version 2021.0 of the H3D writer libraries. This allows Inspire Extrude to avoid writing two H3D files (one is SI and one in user-specified units). However, post-processing will require HyperView version 2021.0 or later.

Metal Extrusion

Enhancements

Filename Length of Particle Trace Files

The length of the seed file names has been increased from 35 to 50 characters.

Billet Skin Computations

Billet skin tracking computations have been enhanced to minimize diffusion and improve accuracy.

Resolved Issues

Billet Preheat in a Multicycle Analysis

Fixed an issue in setting the billet preheat temperature for each cycle. (HXT-391)

Billet Taper Not Applied to Skin Component

Fixed an issue where the billet taper was correctly applied to the billet elements but not to the billet skin elements. (HXT-145).

Billet Skin Issues

- Fixed an issue where the billet skin volume was increasing during the cycle. (HXT-106, HXT-302)

- Fixed a discrepancy between the Skin Volume in the CSV File and the H3D file. (HXT-155)

Numerical Issues in Choke Angle Determination

When the choke angle is determined from a 2D Surface mesh, there was some numerical noise leading to an incorrect choke angle. This was resolved by increasing the tolerance and marking very small/negative choke angles as 0. Additional enhancements were made to improve the accuracy of the captured choke angle. (HXT-248)

Error is Stats CSV File

Fixed a missing comma in the stats.csv file. (HXT-423)

Quenching
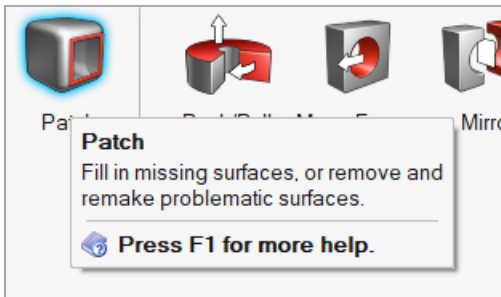
Enhancements

Upper Limit to Grain Size

The quenching solver has implemented an upper limit (maximum grain size) in the grain growth model that can be controlled through data input by the user.
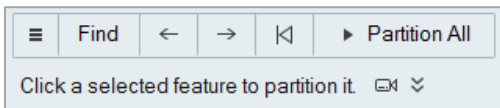
Learn More About Inspire Extrude

You can learn more about new and existing features in Inspire Extrude using the following resources:
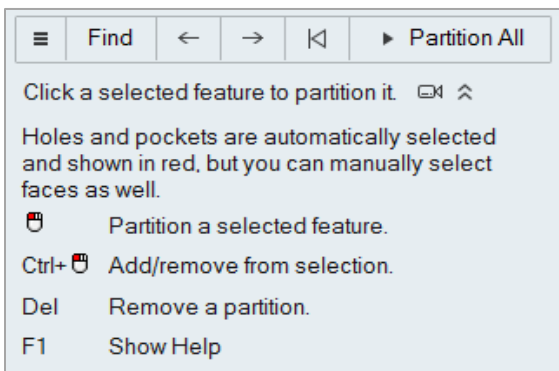
In-Application User Assistance

Inspire Extrude 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.
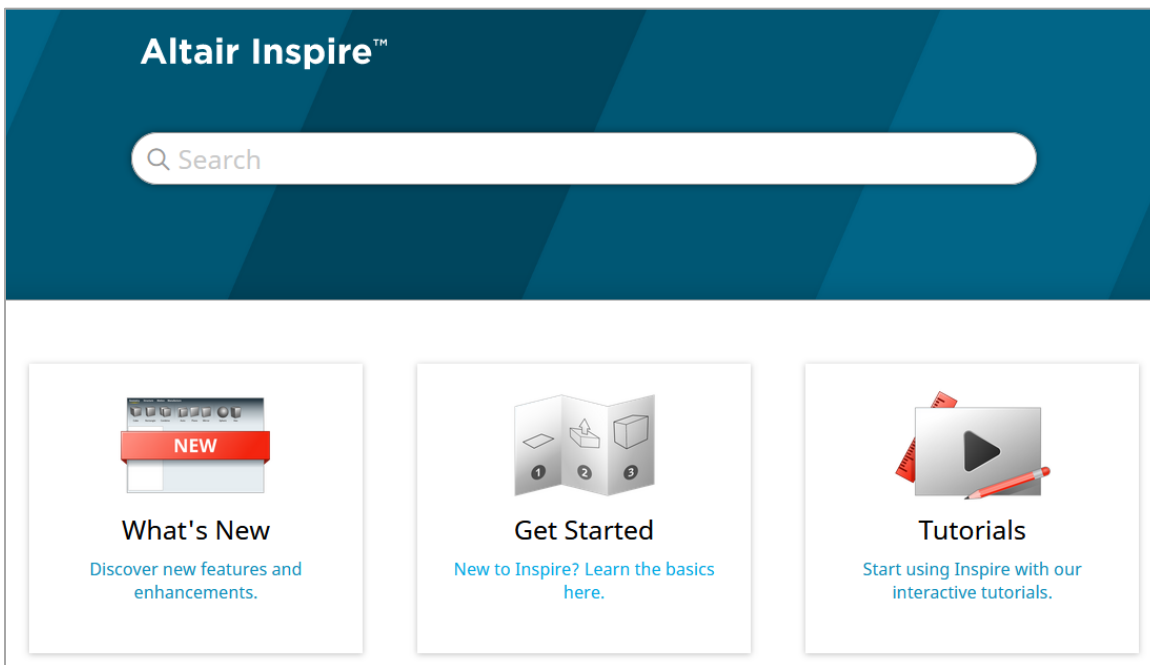

Click to view additional tips and shortcuts. Some tools also include a video .

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.