Altair Inspire™ Studio 2021

New Features and Enhancements

Altair Inspire Studio 2021 includes the following new features and enhancements.

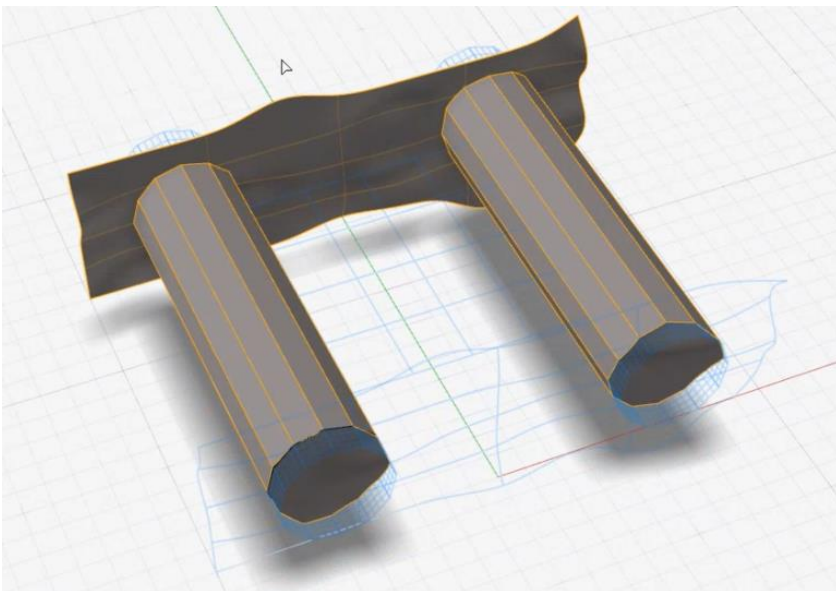## Modeling

Constraint Legend for Sketching

Next to the Control Panel, a legend is displayed indicating by color which sketch curves are Underdefined, Fully Defined, Over Constrained, Fixed, and Imported.

■ Under Defined
■ Fully Defined
■ Over Constrained
■ Fixed
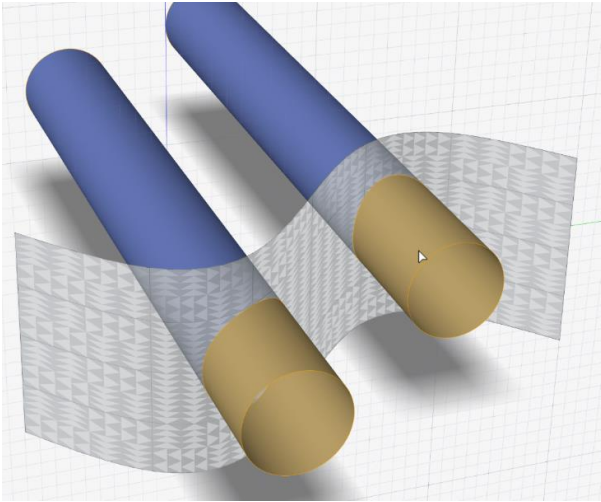■ Imported

New Intersect Tool

The Intersect and Make Manifold tools have been combined into a new Intersect tool with the following features:

- Multi-output depending on the result of intersecting the input objects
- Trimeshes are accepted as input objects
- Automatic creation of solid bodies as output
- Automatic removal of pieces inside solid bodies
- Faces not in solid bodies can be optionally added as individual bodies
- Output can be "combined" or "uncombined" (default)
- Combined result creates one output object with all surface bodies and one trim (if any) object with all facet bodies
- Uncombined result creates one object per solid and one object per additional piece added
- If a body is mixed (made of NURBS and trimesh geometry), it is always divided into two parts

New Split Tool

You can now divide selected surfaces based on a selected trimming tool. You can choose which side to keep or keep both sides.
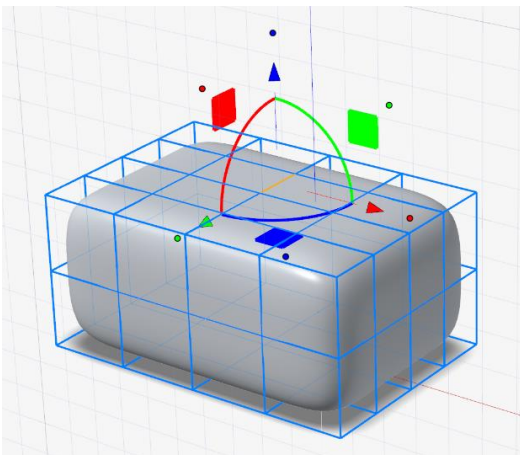

New Push/Push Tool

Use the new Push/Pull tool to modify designs that do not have construction history. There are two modes to Push/Pull: Push Pull and Move Feature. Push Pull allows you to move faces, changing the size or shape of your design. Move Feature allows you to move entire features (such as holes).

New Untrim Tool

Use the new Untrim tool to restore a surface that has been trimmed. Untrim only works on objects without construction history.
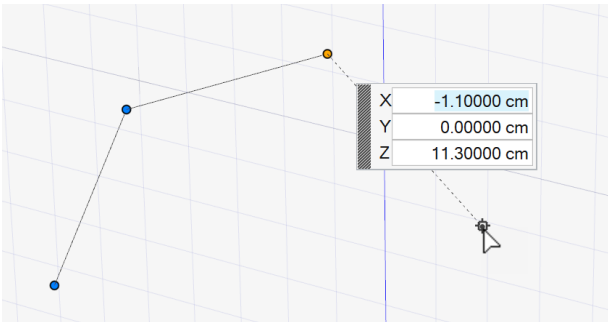
New Mini Move Tool in PolyNURBS

A new Mini Move tool is displayed when editing a PolyNURB. Use the Mini Move tool to move or scale the selected vertices, edges, or faces. For many cases, this eliminates the need to enter and exit the Move command while editing PolyNURBS.
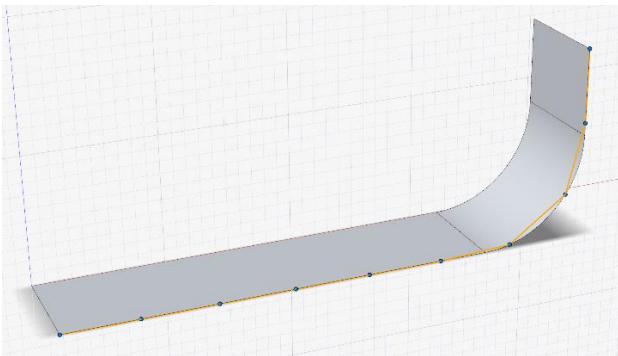
New Create Polyline Tool (PolyNURBS)

You can select points to create a polyline that consists of PolyNURBS edges, and then extrude the polyline to create a PolyNURBS body.
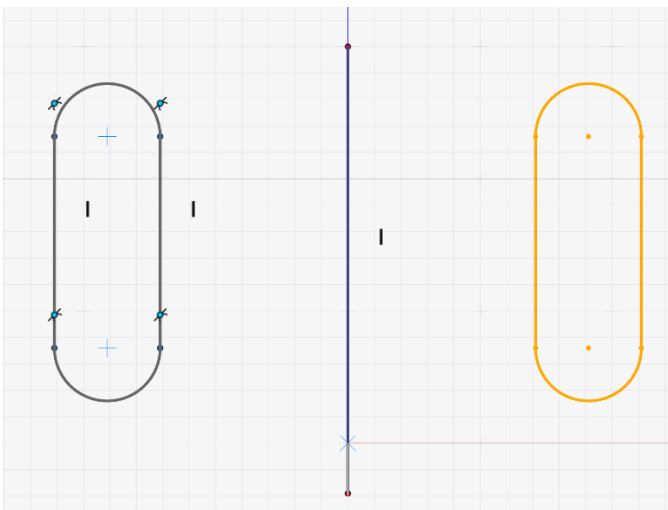

New Extract Polyline Tool (PolyNURBS)

You can select an object to extract the edges as a polyline. The polyline consists of PolyNURBS edges that you can extrude into a PolyNURBS body.
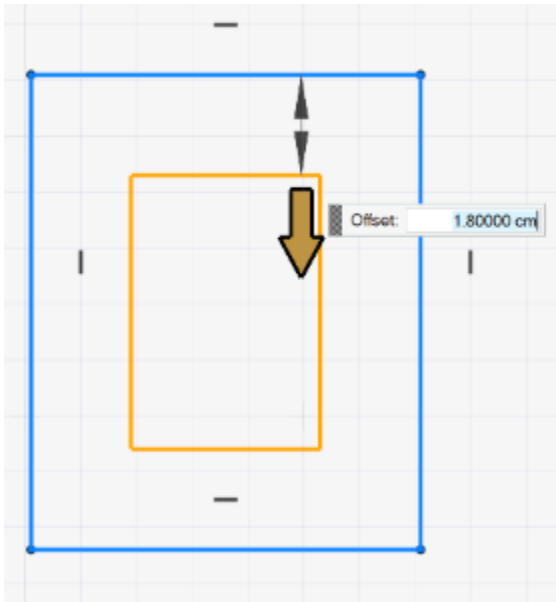

New Mirror Tool (Sketching)

You can mirror selected sketch objects around a selected centerline. The mirrored objects are constrained to maintain their size and position.

New Offset Tool (Sketching)

You can offset selected sketch objects a specified distance. There are options for symmetric offsets and adding caps to the offset geometry.
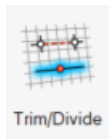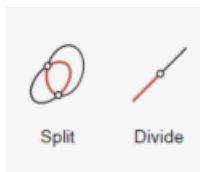

Updates to Design Table

- You can now include sketch dimensions in design tables. Select the sketch in the Browser or modeling window, then select the dimensions to include from the design table.

- You can now create multiple snapshot tables. You can set different values for variables in each table.

Tool Name Changes

- On the Sketching ribbon, the Split tool has been renamed **Divide**.

- On the Curves tab, the Intersect tool has been renamed **Split** and the old Split tool has been renamed **Divide**.

Rendering

Quick Render

You now have options to **Render** and **Render in Darkroom**.

- **Render** will process the image in the application without the overhead of loading Darkroom.

- You then have the option to send the image to Darkroom for processing (**Render in Darkroom**) or save the image.

Die Design

Draw Direction Calculated by Default

When you select a part, the draw direction is calculated by default. You can still adjust the draw direction using the Set Draw Direction tool.

New Symmetry Tool

You can now define symmetry boundary conditions before designing the die face.

Updated Binder Tool

The Binder tool has been updated to support symmetric parts.

Updated Addendum Tool

The Addendum tool has been updated to support symmetric parts.

New Bridge Tool

You can now bridge two addendums or an addendum and a part.
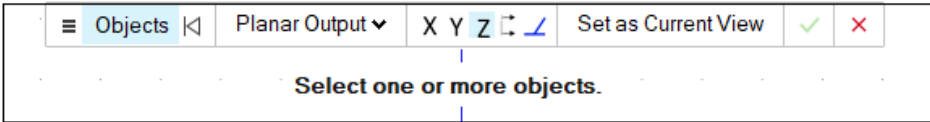
New Matching Die Tool

You can now use the Matching Die tool to create matching dies by offset.

Learn More About Inspire Studio

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

In-Application User Assistance

Prompts appear when you select a tool that opens a guide bar or microdialog. The text prompts you what to do next.

Online Help

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