RELEASE NOTES

Altair Inspire PolyFoam 2021.1

# New Features and Enhancements 2021.1

Release Highlights

The Inspire PolyFoam 2021.1 release features new and improved tools for sketching, geometry, and PolyNURBS, as well as four new result types.

New and Improved 2D Sketching

Inspire PolyFoam 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.
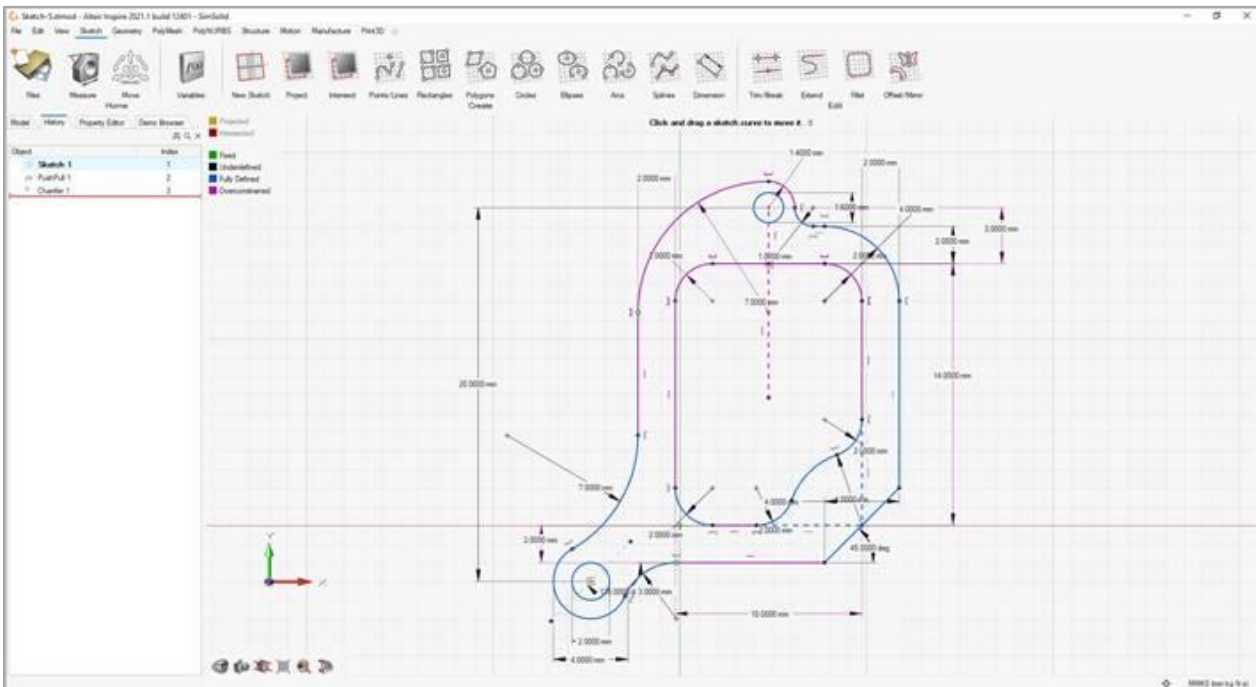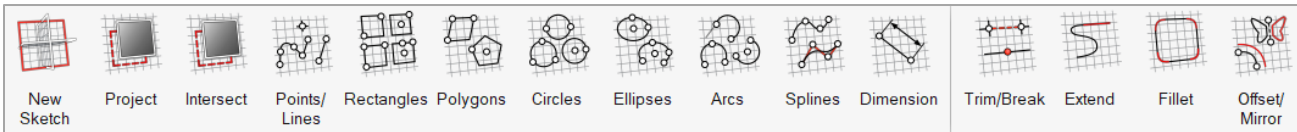
New Geometry Modeling Tools

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

| Reference	Push/Pull	Extract	Mirror	Patch	Move Faces	Scale	Boolean	Slice	Simplify	Partition	Midsurface	Fillets	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.

Create	Fit	Wrap	Pave	Move Bodies	Mirror Bodies	+/-	Split	Bridge	Sharpen	Repair	Adjust	Shape

Global References

Inspire PolyFoam 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.
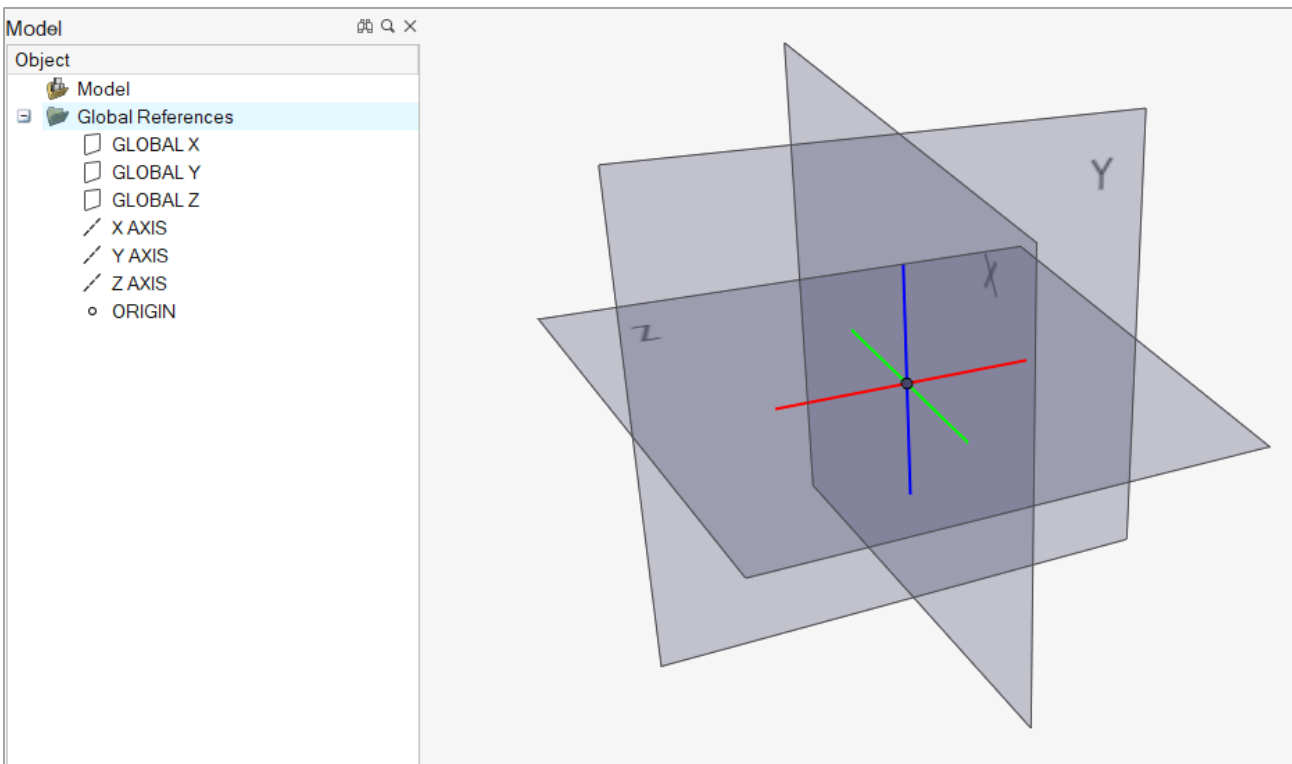
New Result Types

Four new polyfoam result types for isocyanate, viscosity, surface defect modulus, shrinkage defect modulus have been added in the Analysis Explorer.

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.
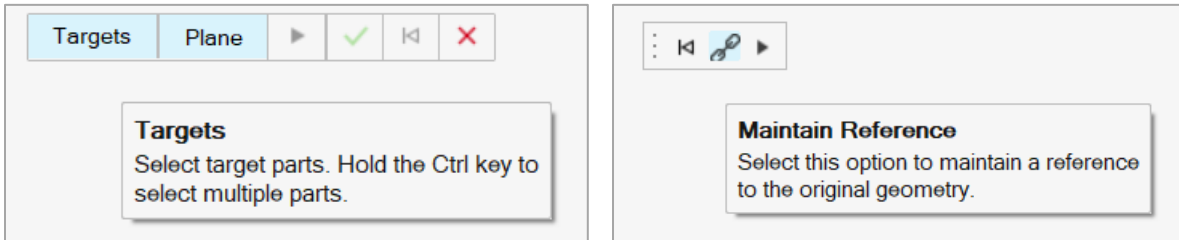
Sketching

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).
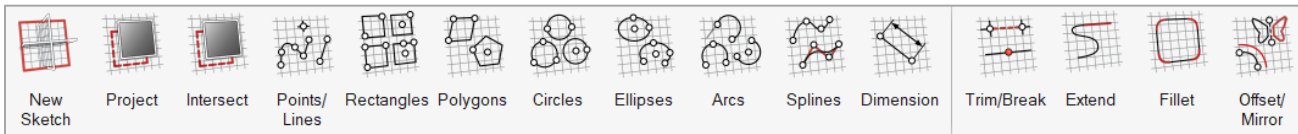
Constraints Legend

Inspire PolyFoam 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.

PolyFoam

Inspire PolyFoam has added four new result types for the 2021.1 release, which are available in the Analysis Explorer.

Isocyanate and Viscosity Result Types

It is now possible to view the viscosity and isocyanate remaining during the injection, foaming, and curing stages of the simulation.

- **Isocyanate**: This result type simulates remaining or unreacted content at the end of the stage. The residual content of polyol and isocyanate is a good indicator of chemical properties.
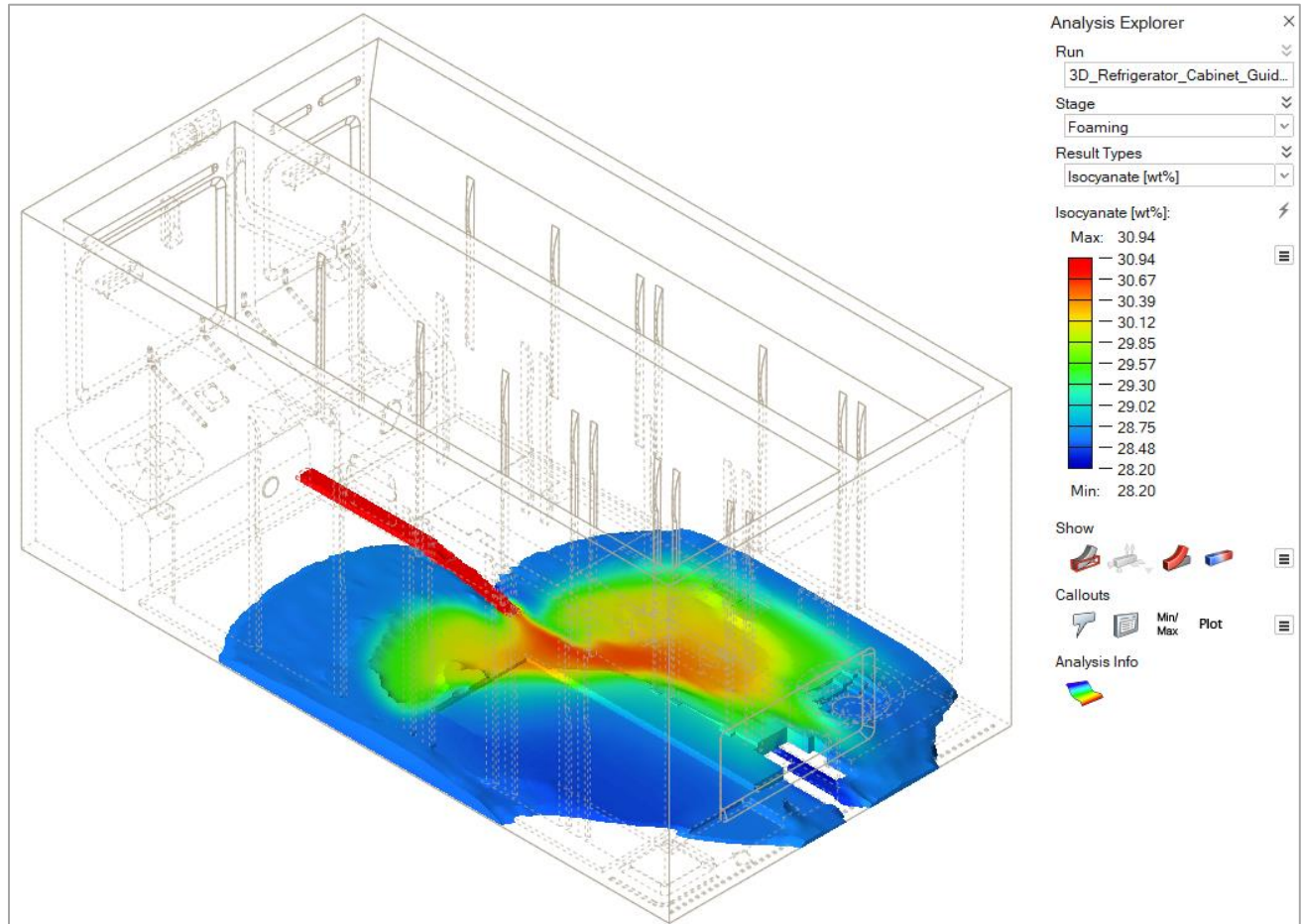

- **Viscosity**: This result type shows the change in viscosity during the stage, which depends on the chemical reaction and fluid flow.

Surface and Shrinkage Defect Results

In addition to air pressure and filling time, you can now view potential surface and shrinkage defects at the end of the foaming process.

- **Surface defect modulus**: This result type predicts surface defects such as coarse or tough areas by using the shear force. It depends on the mold temperature, pouring mass, and shear rate, and can be used for all kinds of polyurethane foam.

(a)   (b)   (c)
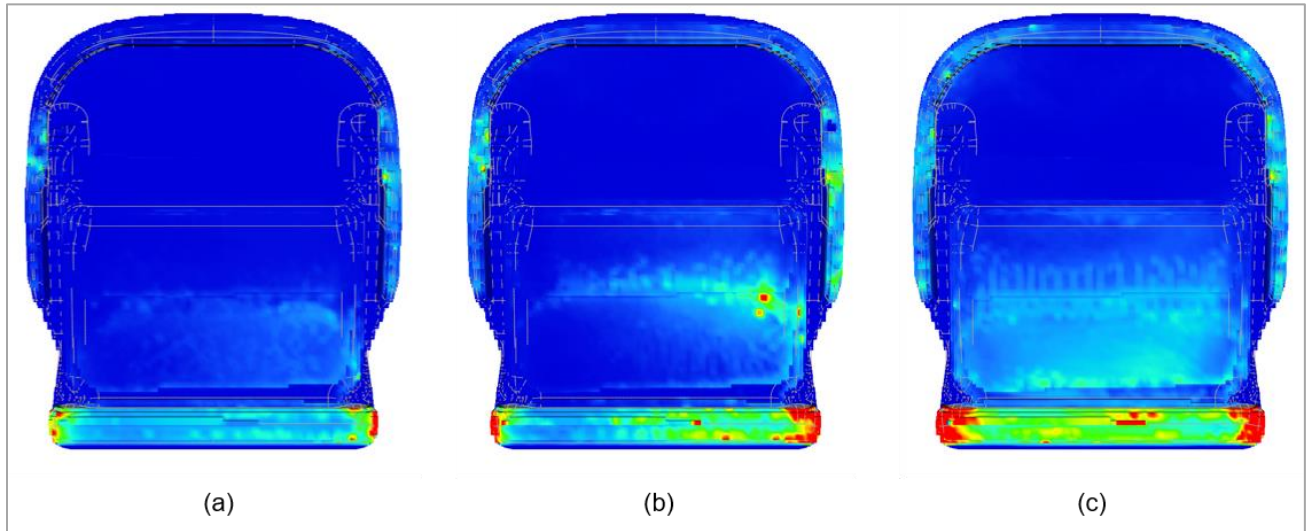
- **Shrinkage defect modulus**: This result type predicts volume shrinkage of the foam, which occurs if the foam is too soft even after demolding. It depends on the demolding time, pouring mass, mold temperature, and reaction rate, and can only be used on flexible polyurethane foam.

(a)   (b)

(c)   (d)
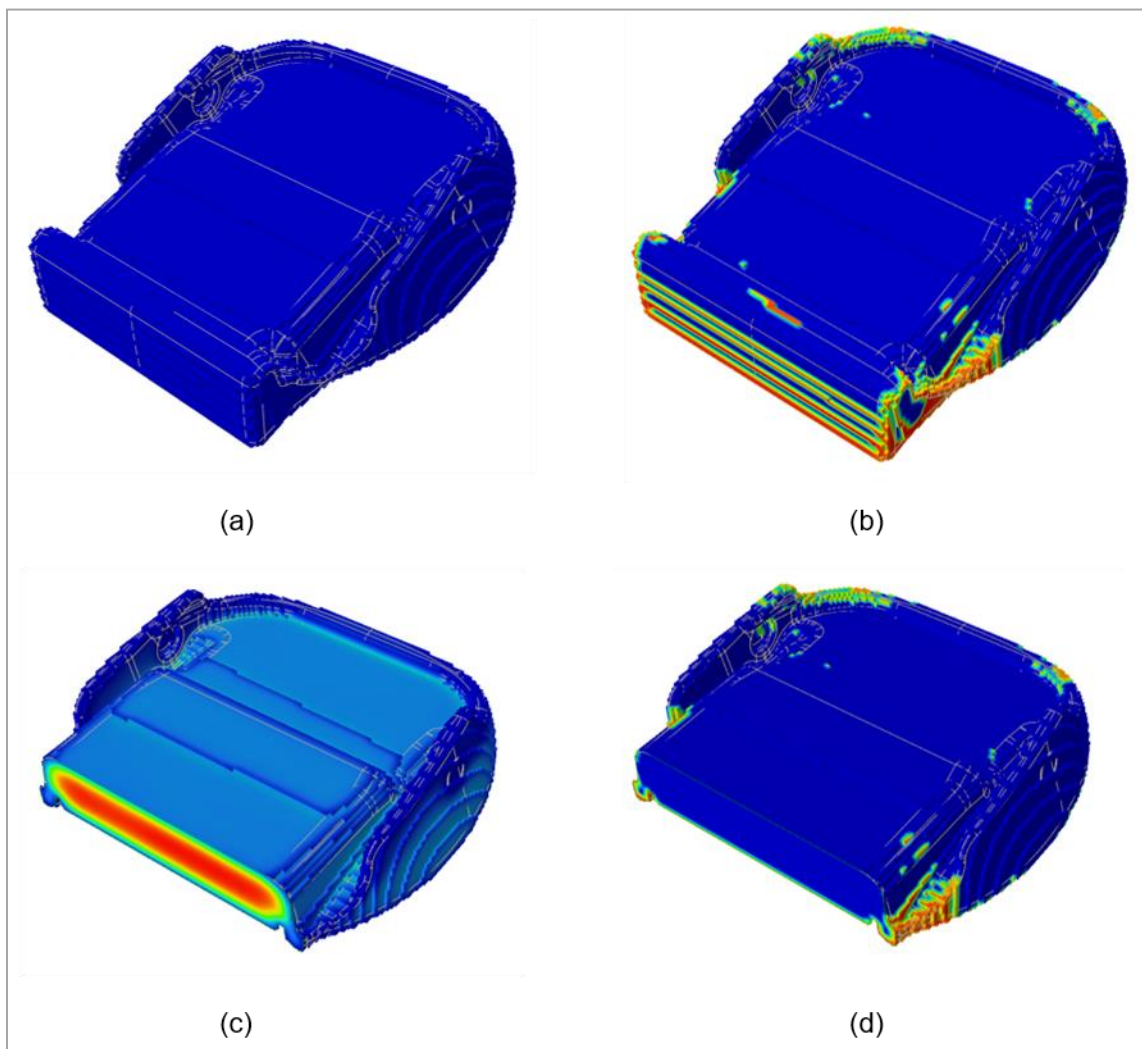
Remove Isolated Elements

Meshing has been improved to automatically remove isolated and disconnected mesh elements in curved and thin areas.
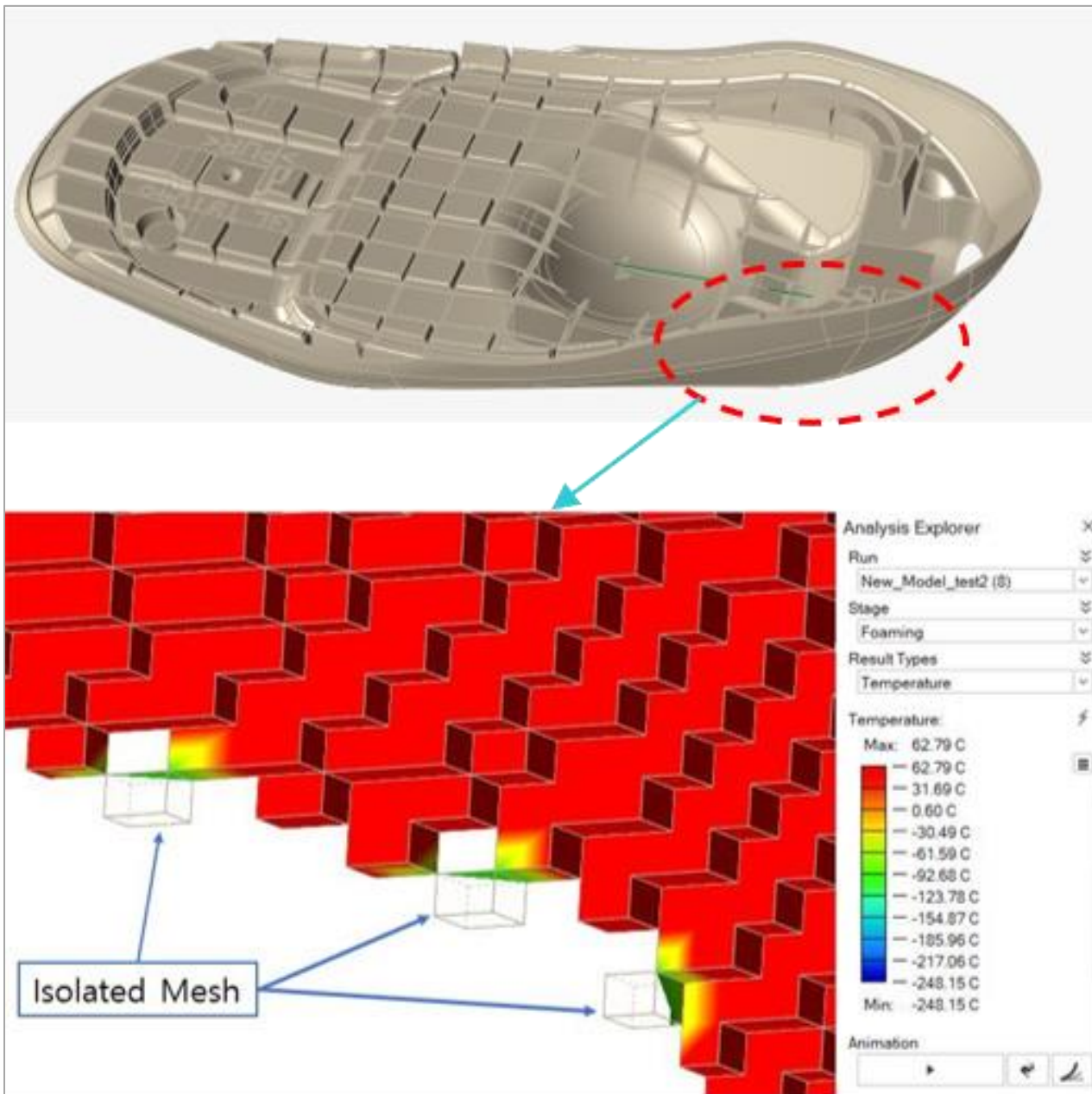
Learn More About Inspire PolyFoam

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

In-Application User Assistance

Inspire PolyFoam provides two types of user assistance. **Enhanced tooltips** appear when you hover over icons and other features. They describe what the tool does.
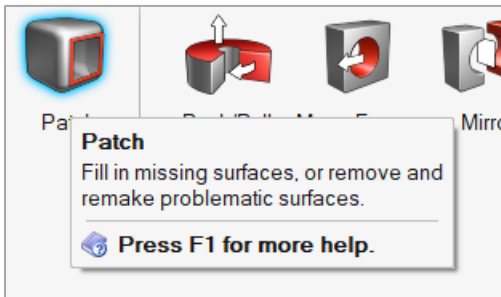
Workflow help appears when you select a tool that opens a guide bar or microdialog. The text prompts you what to do next.
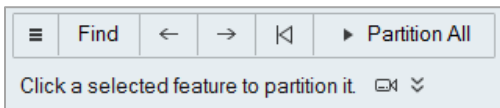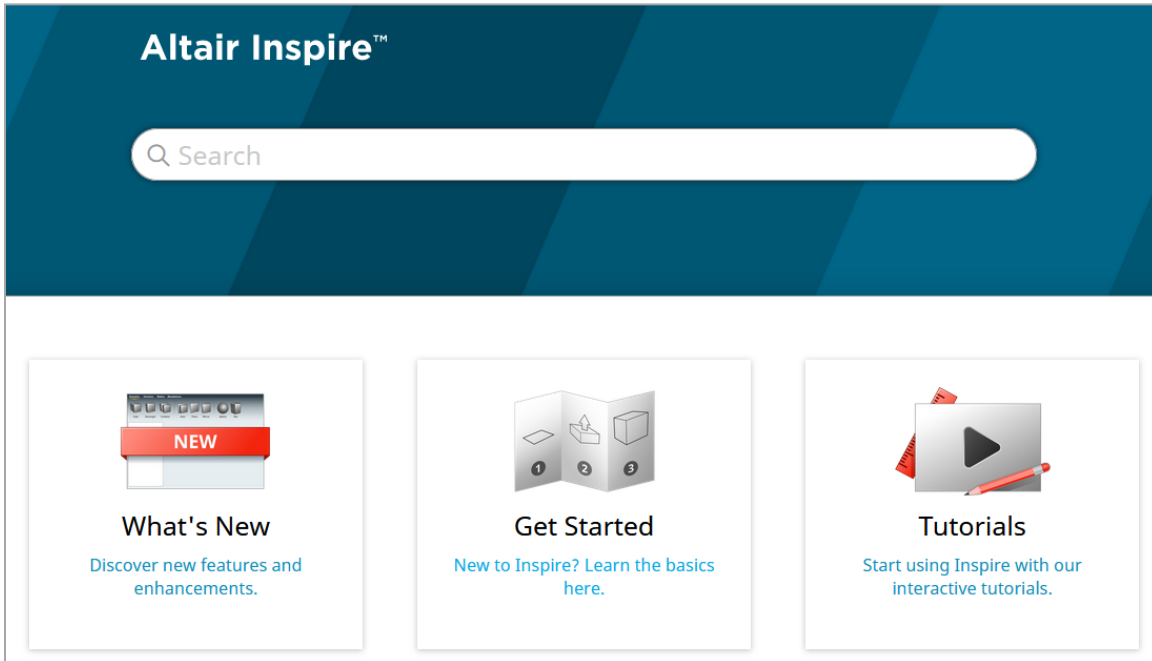

Click ⌄ to view additional tips and shortcuts. Some tools also include a video ⌧.

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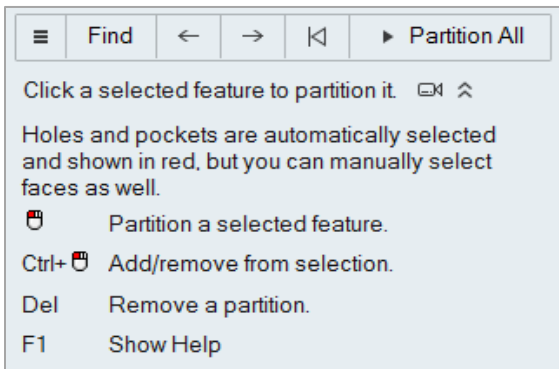
Online and Offline Help

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.