Altair® Inspire™ PolyFoam

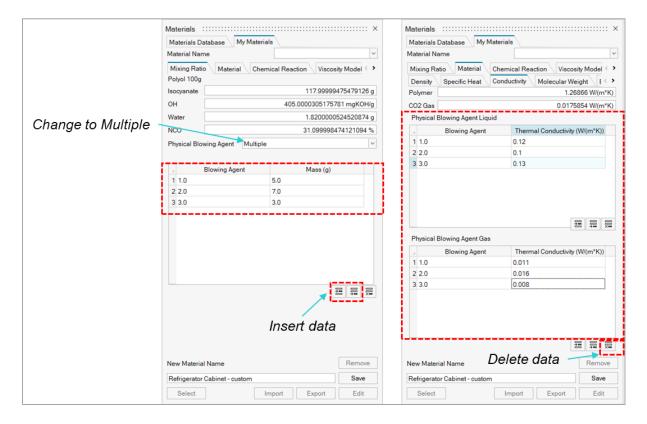# 2021.2

New Features and Enhancements 2021.2

Altair Inspire PolyFoam 2021.2 includes the following new features and enhancements.

Foaming

Multi-Physical blowing agent (PBA)

The actual blowing process of rigid PU foam is the physical blowing process in which a low-boiling liquid incorporated in the reaction mix is vaporized due to the heat of the reaction. Various PBA is used together or alone for responding to global climate change and good properties such as low-temperature foaming and low thermal conductivity. Since only one type of PBA could be set in the previous version, a new feature was added to enable multiple PBAs for user convenience in this version.

Users can set the number of physical blowing agents in the Mixing ratio tab of Materials window like the figure below. Material properties such as density, specific heat, conductivity, boiling temperature, and molecular weight need to be input as many as the number of PBA.

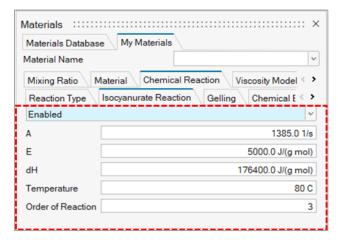### Isocyanurate reaction (PIR)

Analysis results now include an isocyanurate reaction result type. Poly-isocyanurate (PIR) foam is recognized as the efficient thermo-insulations utilized for constructions. Manufacturing processes of PIR foams are like those of rigid polyurethane (PUR) foams. Isocyanurate linkages are formed by cyclotrimerization of excess isocyanate compounds using suitable trimerization catalysts. The outstanding properties of PIR foams are inherent thermal stability and low flammability.

PIR reaction can be described by the following formula. There is a very complex reaction mechanism, but a simplified reaction equation is used for modeling.

$$3[NCO] = [PIR]$$

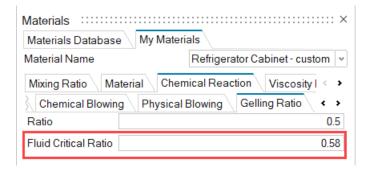$$\frac{d[PIR]}{dt} = A_{PIR}exp\left(-\frac{E_{PIR}}{RT}\right)[NCO]^n$$

The parameter input window of the kinetic equation is composed as follows, and the meaning of each is described below.

- A: Pre-exponential factor

- E: Activation energy of reaction

- dH: Heat of reaction

- Temperature: Reaction start temperature

- Order of reaction: Order of reaction in reaction kinetics (n)

Fluid critical gelling ratio

Fluid critical gelling ratio means the critical(limit) gelling ratio at which flow can occur. When the gelling ratio reaches this value, each element can no longer flow and does not foam, as if it had solidified. It is implemented for smooth rise at the final stage.
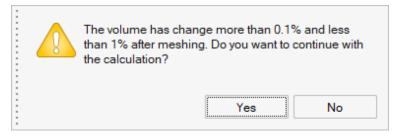

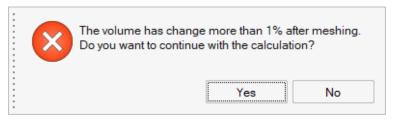# Additional Changes and Enhancements for Foaming

- **Mesh Volume Change Warning**: Inspire PolyFoam now calls attention to removed mesh volume. A relatively small element in which the voxel mesh is fragmented or separated from the product part is automatically deleted, and Inspire PolyFoam informs the user. Remeshing may be necessary if the volume removed is greater than 1%.
  - If Removed Volume% == 0: Nothing is shown
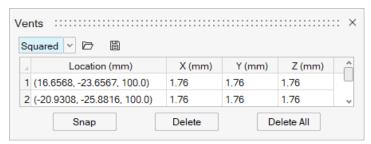 - If Removed Volume% <= 0.1%: Informational dialog appears

  > **i** The volume has change less than 0.1% after meshing.
  >
  > OK

  - If 0.1< Removed Volume% <= 1%: Confirmation dialog appears. User can continue or stop the calculation

  > ⚠ The volume has change more than 0.1% and less than 1% after meshing. Do you want to continue with the calculation?
  >
  > Yes   No

  - If Removed Volume% > 1: Confirmation dialog appears. User can continue or stop the calculation.

  > ❌ The volume has change more than 1% after meshing. Do you want to continue with the calculation?
  >
  > Yes   No

- **Prefill Mass and Smooth Mesh for Pouring Cup Test**: When analyzing the cup model for D/B, a function was added so that the amount of the PU solution can be entered as a mass to improve user convenience. Also, the Smooth Mesh option in Run Analysis is turned on automatically for cup test to ensure the accuracy of the results.

- **Square Vents Table**: Square vents can now be viewed in a table.

Location (mm)	X (mm)	Y (mm)	Z (mm)
1 (16.6568, -23.6567, 100.0)	1.76	1.76	1.76
2 (-20.9308, -25.8816, 100.0)	1.76	1.76	1.76

Snap   Delete   Delete All

Resolved Issues

- **Analysis showed the same results after changing the initial temperature of the foam part**: This issue has been resolved, and now changes to the initial temperature of the foam part will affect the analysis results.
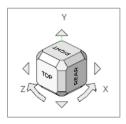
- **PU disappeared in thin-layered areas when foaming starts**: The mass balancing algorithm has been upgraded to preserve mass conservation.

- **Excessive analysis calculation time under certain conditions**: The algorithm for movement and correction of fluid has been upgraded.

- **Air pockets in PU foam caused solver to crash**: The code has been upgraded to prevent crashing under these conditions.

- **The tab label "Foam Thixo Model" in the Materials window was unclear to many users**: Changed the tab label to "Foam Rheology Model" for clarity.

- **'Run at model location' option did not work**: This issue is fixed.

- **The nozzle table showed only the first line**: This visualization issue is fixed.

Geometry

View Cube

A new navigation cube has been added to the view controls in the lower left corner of the modeling window, replacing the old View Rotator.

- Clicking a major face (Top, Bottom, Right, Left, Rear, or Front) rotates the model to the nearest possible orientation of that face. If that orientation is not the standard, clicking the face again re-aligns the model back to the standard orientation. If the orientation is already standard, repeated clicks on a major face reverses the view.

- Clicking the arrows incrementally rotates the view. Left- and right-clicking increments by + and - 15 degrees while middle-clicking increments by 90 degrees.

- The display and behavior of the View Cube can be adjusted in the Preferences under **File→Preferences→Inspire→Visualization**.
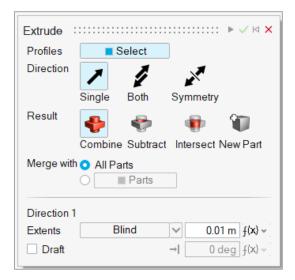
Extrude Tool

Use the Extrude tool to sketch profiles in one or two directions. Create a new part or combine, subtract, or intersect the extruded shape with existing parts.

In the guide panel, select profiles and choose a direction, result type, and merge method.
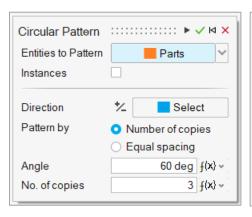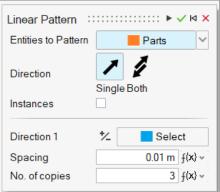
Pattern Tool

Use the Pattern tools to create a linear or circular pattern of parts or faces.

In the guide panel, select the entities to pattern and define the direction.
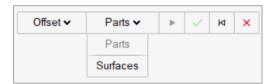

Offset Tool

Use the Offset tool on the Geometry ribbon to offset parts or surfaces.

Choose the Parts or Surfaces on the guide bar, then select either parts or surfaces in the modeling window.

5

You can also thicken parts by selecting Thicken on the guide bar. Surfaces cannot be thickened.

Additional Changes and Enhancements for Geometry

- The sketch Intersect tool can now be applied to parts or surfaces.

- Inspire now reads surface color meta data from Catia files.
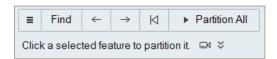
Learn More About Inspire

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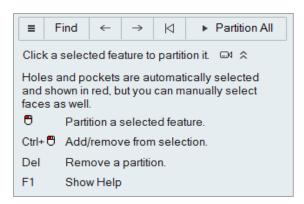
In-Application User Assistance

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.
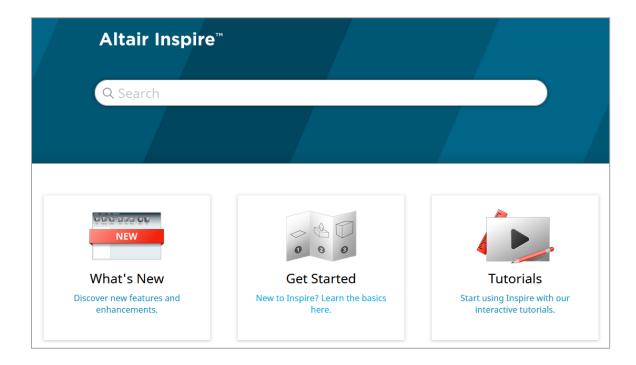

Click to view additional tips and shortcuts. Some tools also include a video .

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.