RELEASE NOTES

Altair® Inspire™ Extrude 2022.1

New Features and Enhancements 2022.1

The Inspire Extrude 2022.1 release includes the following new features and enhancements.

General

Parametric Modeling – Construction History

All Inspire Extrude interfaces are upgraded to enable construction history. Parametric modeling captures the evolution of your model's shape by showing the linear sequence in which the geometry was created. You can roll the model backward or forward, as well as group, delete, insert, suppress, or edit entities in the History Browser. When you make a change, it is propagated through the sequence of operations, and the model is updated automatically. The entities in the History Browser are called "construction features" and they are interdependent.

In this release, the recommendation is to use the current workflows for model setup and consider parametric modeling an experimental feature. The main motivation of this parametric modeling is to support parametrized die design modules in future releases. (IEXT-2759)

Resolved Translation Issue

In the previous release, the results list was not translated to the preferred language in the results explorer dialog. This issue is resolved in this release. (IEXT-2767)

Metal Extrusion

Choke Parameters

In this release, analysis with *bearing reference surfaces* is enhanced by the addition of a new parameter to specify the maximum choke angle in the export dialog. This parameter sets the bound for the maximum choke angle determined from the reference surface. (IEXT-2741)

Improvement in Organize

In this release, re-organizing the extrusion parts is made easy. In the previous releases, users could perform *organizing parts* only once. If any corrections to the part size are needed the FlowVolume back. Now, users can change the part sizes if required again by just clicking on the Organize GUI. (IEXT-2767)

Improvements to Bearing Optimization Results

The solver is improved to add more details and a summary of bearing optimization runs and results. As a part of this enhancement, a new CSV summarizing the results is created. The OUT file now includes additional details about the bearing optimization run and iterations. (HXT-175)

Resolved Issues

The following issues have been resolved for the 2022.1 release:

Metal Extrusion:

- Bearing Optimization runs all 20 iterations. This is not an issue as the solver looks for a strong convergence and expects every bearing control point to be within the specified tolerance. However, to improve computational efficiency, now the solver uses

RMS-based criteria for determining convergence. If the old behavior of a stronger check is needed, you can turn it on by setting the parameter *BOStrongConvergenceCheck* to yes. The default value of this parameter is no. (HXT-245)

- Variable time steps sometimes lead to a solver crash. The solver computes the time step using the variable time step command specified in the input file. Due to an error in implementation which was triggered by rounding off issues, especially the zonal transitions, the time steps were estimated incorrectly at these transitions. This was leading to excessive consumption of the billet during the cycle and hence, a solver crash. The resolution of this issue is applicable only for a single billet cycle or multiple billet cycles with the same billet length in every cycle. (HXT-490)

- Solver crashes with error AVERAMPRESS when the press is selected. The solver no longer throws errors while computing average ram pressure for Metal extrusion problems. (HXT-511)

- Error in the CSV files. Minor errors from missing commas in the CSV file, which were making files unusable, are now corrected. (HXT-473, HXT-479, HXT-487)

Quenching

- Immersion quenching results diverge from experimental results. The implementation for the heat transfer coefficient computation was enhanced for the immersion quench. Now the solver uses various correlations to model the different regimes of the boiling curve, which includes the nucleate boiling and the membrane (film) boiling. (HXT-523)

- Air fan quenching not working in version 2022. The solver now uses the computed values of heat transfer coefficients, so temperature change due to air fan quenching is correctly calculated. (HXT-516)

- Center walls cool faster than the outer walls exposed to air fans. All the surfaces in an air fan quench zone were getting quenched even if the surfaces were not directly exposed to the air fans. The solver is now enhanced to consider visibility calculations for computing the cooling effect of an air fan. This helps to treat the surfaces in the quench zone which are exposed to air fans differently from those which are not directly exposed to air fans. Also, now the solver ignores the fan cooling at the points which are outside the impingement width of the fan. (HXT-517)

Polymer Extrusion:

- Viscoelastic model material data. The issues in writing the CEF Viscoelastic model data to the MTL file are resolved. (IEXT-2781)

New Features and Enhancements 2022

The Inspire Extrude 2022 release includes the following new features and enhancements.

General

### User-Controlled Color Codes for Extrude Components

The colors used for different metal extrusion components such as billets, dies, etc. can now be controlled in the Preferences. Using a standardized color scheme for the components enables easy identification and enforces a standard within a company. (IEXT-2615)

Mesh Control Options

You can now control the mesh size of different parts from the Run Analysis dialog. This option is provided under Mesh Size and is in addition to the current options: Coarse, Medium, Fine, and Existing Mesh. This enables you to make the mesh coarse or fine, as desired, in different regions. (IEXT-2604)

Dark Theme

Dark Theme is now supported throughout Inspire. You can change the theme in the **Preferences** under **Workspace > Theme**.
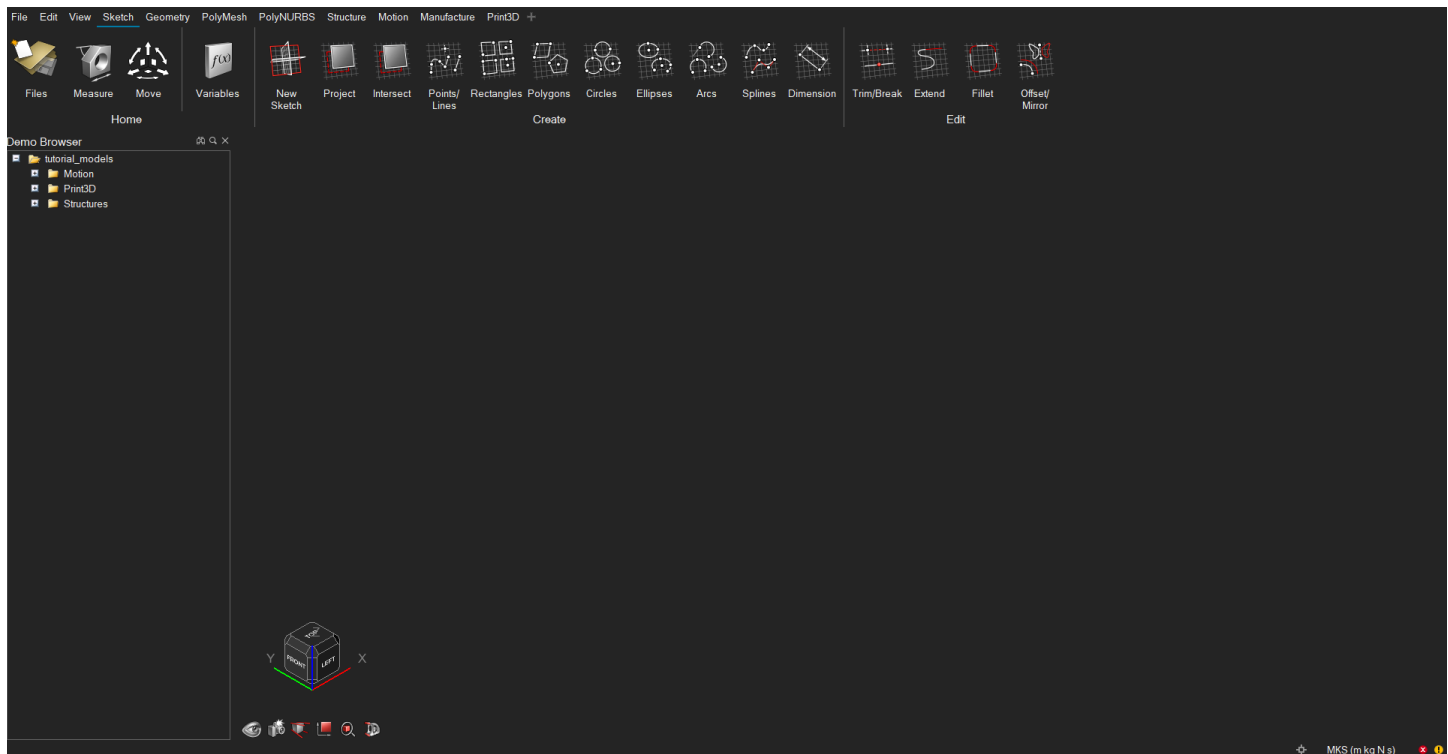
Advanced Screen Capture

Several new features have been added to the settings for the Advanced Screen Capture feature available in the File menu. These include the ability to copy an image to the clipboard, as well as options to either show a file save dialog or save files to a default location. These options can also be accessed from the Preferences.
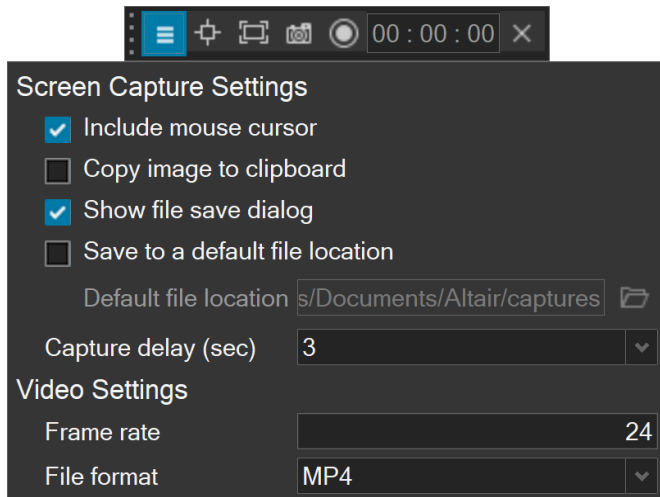
Geometry

Keyboard Shortcuts for Sketching

When outside of sketch editing mode, you can now press the **S** key and select a face to create a new sketch. Once in sketch mode, you can quickly access the most commonly used sketch tools with the following shortcut keys:

Hotkey	Tool/Command
A	Arc by Center and End Points
B	Trim
C	Circle by Center and Point
D	Dimension
G	Turn Grid On/Off
K	Fillet
L	Polyline (press Shift to toggle between lines and arcs)
M	Mirror
O	Offset
R	Rectangle by Corners
S	Create New Sketch (when not in sketch mode) Spline Using Control Points (when in sketch mode)

Press **Esc** to exit any sketch tool.

New Pipe Tool

You can now turn lines and edges into a pipe. The pipe has a uniform cross section that can be circular, square, rectangular, or triangular.

New Slice Tool

In addition to slicing a part with a cutting plane, you can now slice a part with a surface. The surface must at least partially intersect the part.

New Intersect Surfaces Tool

You can now retain only the intersecting portions of one or more parts. This is a "smart" intersect tool that automatically deletes excess faces, edges, and vertices.

Extrude Tool Updates

The Extrude tool has been enhanced for this release. You can now extrude all types of profiles, as well as 2D surfaces in one or two directions, to create solids or 3D surfaces with optional end caps. In addition, you can now extrude in a custom direction by either (1) selecting a reference geometry to which you want to align the extruded shape or (2) freely orienting the extruded shape by dragging the curved arrow or entering an angle in the microdialog.

Pattern Tool Updates

The Pattern tool has been enhanced and now allows you to perform Boolean operations on the results, as well as merging with all or selected parts. In addition, the new Copies to Skip option allows you to select copies to exclude from the pattern. For linear patterns, selecting the Seed Only checkbox will restrict the pattern to the first row and column.

Draft Tool Updates

You can now add draft to one or more faces of a part when designing a product that is manufactured using injection molding.

Additional Changes and Enhancements for Geometry

- The workflow for the Boolean Combine tool has been updated.

- Performance improvements for import of CAD files.

- Inspire now supports the following file format versions for import:

Format	Versions
ACIS	All -> R27
Catia V4	All 4.xx
CatiaV5	R10 -> R31
IGES	5.2 & 5.3
Inventor	All -> 2021
JtOpen	All -> 11.0
NX	11.1 -> CR 2007
Parasolid	All -> 33.1
ProE	13 - Creo 8
Solidworks	99 -> 2022
STEP	203/214/242

Metal Extrusion

Heat Transfer Subcase in Coupled OptiStruct Analysis

Inspire Extrude supports coupled analysis with the OptiStruct solver. When the pressure in the extrusion simulation reaches the specified convergence tolerance, the solver writes the loads on the contact surfaces between the workpiece and the tool. These loads are used by OptiStruct to perform the tool stress analysis and compute tool deformation. The modulus of the tool varies significantly with temperature. To consider this, the coupled solution has been enhanced to perform a heat transfer step subcase before performing the stress analysis. To enable this, the solver writes temperature on the tool contact surfaces in addition to pressure. This new feature increases the accuracy of the stress analysis solution. (IEXT-2656)

Advanced Tool Heat Transfer Analysis

You can now perform advanced tool heat transfer analysis using the HyperXtrude solver itself for both profile nose cone and full-cycle analysis. With this new feature, you can access the tool thermal management system supported by the solver. In future releases, all the solver options will be made available to users. (IEXT-2030)

Mandrel Offset

The recommended procedure is to flush the mandrel and remove the offset before creating the mesh. This is accurate and also will result in good quality mesh in the welding chamber region. The solver also supports the scenario where the user can mesh the model with the mandrel offset intact and not flush it at the CAD/mesh. The solver will account for this in the computation by the specified value of the mandrel offset. This helps save time in correcting and updating the geometry. In this release, Inspire will support the specification of mandrel offset in the GUI. This option is shown in **Advanced Bearing Parameters** and should be specified only for the bearing curves on the mandrel. (IEXT-2375)

Reorient the Model After Simulation

Inspire orients the model such that the profile comes in the positive Z direction. In this release, an option to reorient the model to the original configuration has been implemented. This is mainly intended for bearing optimization analysis to get the optimized bearing curves in the original configuration. This feature will work only if you have set up the model in version 2022. (IEXT-2045)

Improvements to Choke Angle Computation

The solver uses bearing reference surfaces to compute the choke angle in the bearing region. An additional mechanism has been implemented to suppress the noise in the choke angle due to geometric inaccuracies in the underlying reference surface. This improves the choke angle estimations from the bearing reference surface.

Resolved Issues

- Fixed issues with imported bearing curves. When the user imports bearing curves generated outside Inspire using other CAD software, sometimes they are generated in a local coordinate system. When this happened, it was causing failures in the model setup. (IEXT-2621 & IEXT-2654)

- Fixed an issue where updated press/material not considered in process optimization. In previous releases, process optimization was computed once after the model is set up and to save computational time it was reused for that model. Because of this, any changes to the press or material database after the process optimization was performed the first time were not considered. (IEXT-2627 & IEXT-2690)

- Fixed issues with post-processing issues for full cycle analysis. When the user tries to post-process a full cycle analysis in the same instance of Inspire after it was used for profile nose cone analysis, the results were not loaded correctly for all the time steps. (IEXT-2597)

- Fixed an issue with Loads on Workpiece—Tool Contact Surfaces. The solver writes the computed extrusion loads on the workpiece—tool contact surfaces to perform stress analysis with OptiStruct. This export was not working for a transient full-cycle simulation. (HXT-485)

Polymer

Resolved Issues

- Fixed a material creation issue where materials created in Inspire were coming with the viscoelastic model enabled to CEF even when it was set to none. (IEXT-2610)

- Fixed an issue when customizing viscosity plots. In the material database, users can visualize the viscosity data that is a function of shear rate and temperature as plots. When the range of shear rate used for the plots was adjusted in the GUI, it was leading to an application crash.

Quenching

Improvements to Heat Transfer Computations in Immersion Quenching

The solver uses more accurate heat transfer correlations to model the phases of the cooling curve for Immersion Quench. Additional parameters have been added to modulate the cooling curve. The solver also accounts for the effect of subcooling on the critical heat flux.

Natural Convection HTC on Free Surfaces Outside the Quench Zone

The solver automatically computes the heat transfer inside the quench zone. Inspire users can now specify the natural convection heat transfer coefficient used for the regions of the profile that are outside the quench zone. In the previous release, a default value was used by the solver. (IEXT-2075)

Resolved Issues

- Resolved quench system visualization issues. When models containing quench systems were reopened, sometimes the graphics were not rendering properly. These graphics issues related to the quenching system were resolved.(IEXT-2680 & IEXT-2678)

# Resin Transfer Molding

Layered Mesh Options

Inspire Resin Transfer Molding now provides options to apply a layered tetrahedral mesh and control the number of layers. This helps create better meshes for the analysis. (IEXT-2657)

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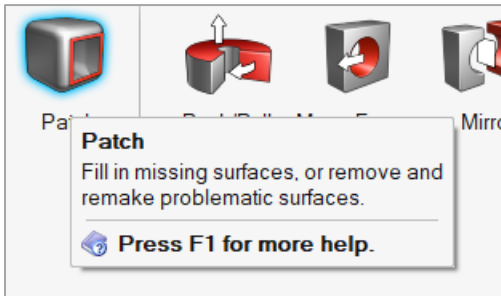
Workflow help appears when you select a tool that opens a guide bar or microdialog. The text prompts you what to do next.
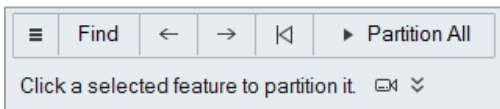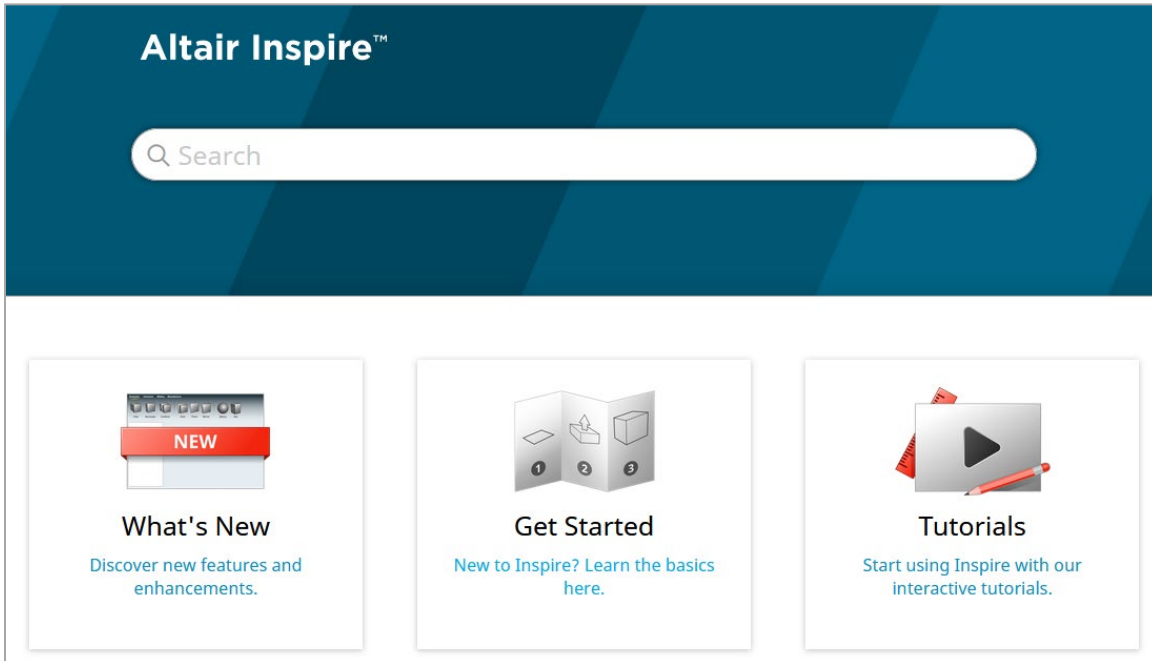

Click to view additional tips and shortcuts. Some tools also include a video .
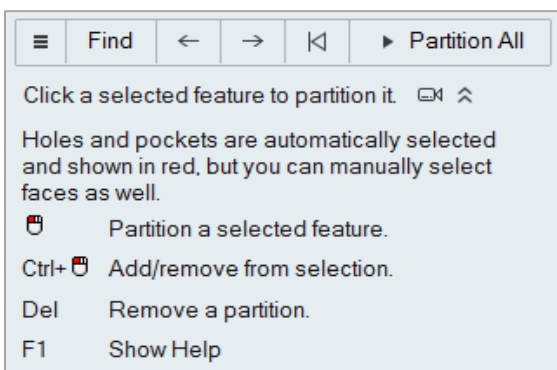
Online and Offline Help

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.