

 RELEASE NOTES

Altair[®] Inspire[™] Form 2022 and 2022.0.1

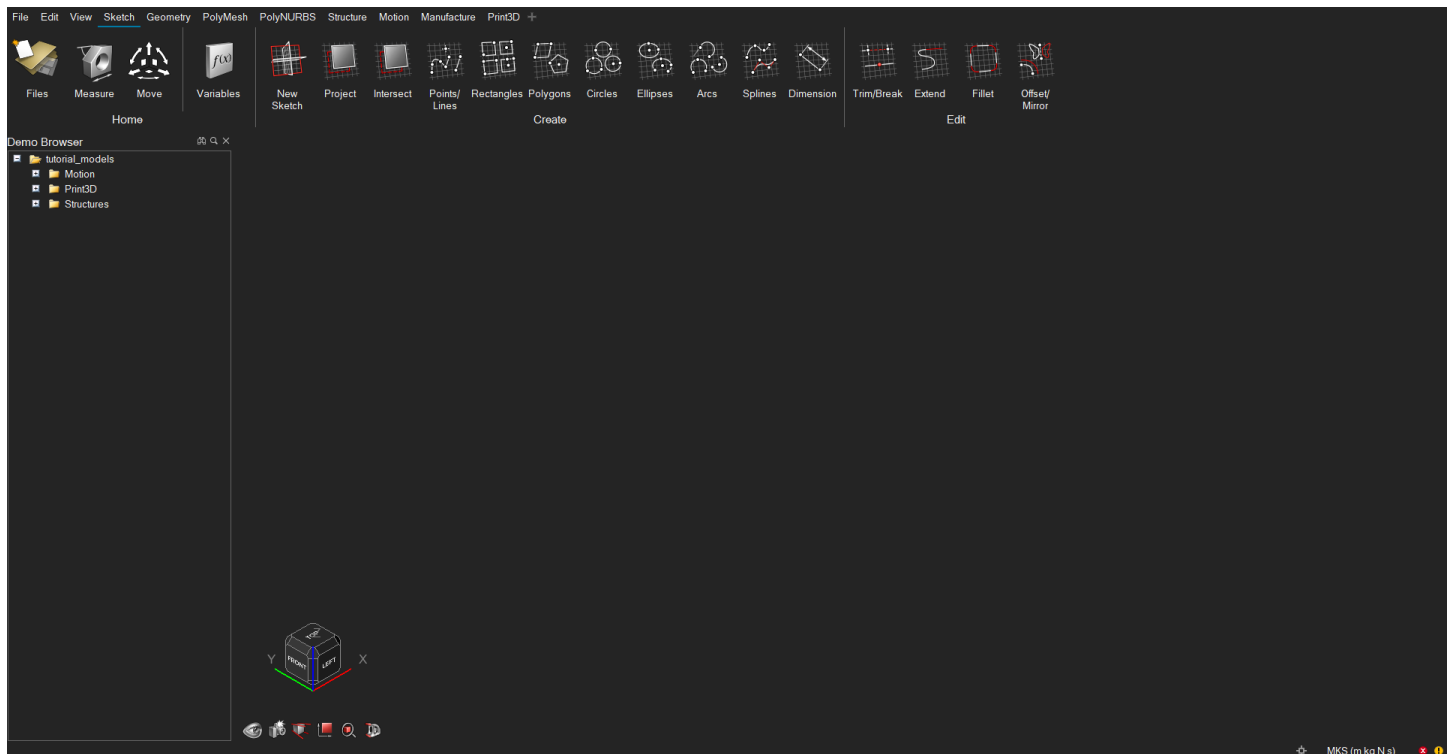
New Features and Enhancements 2022

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

General

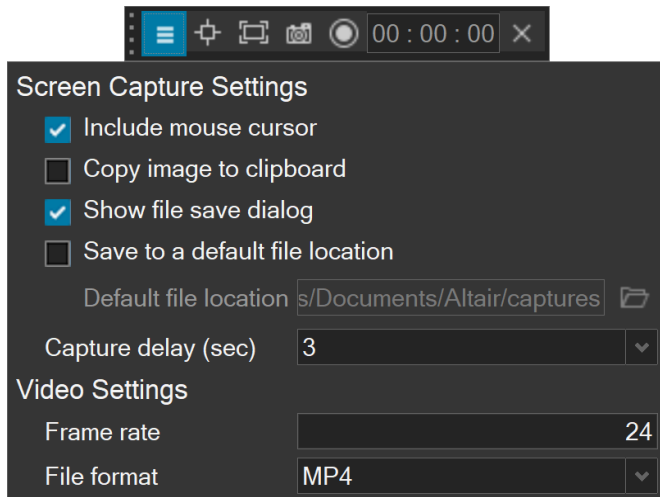
Dark Theme

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

Feasibility

New Extend Tool

Extend surface edges naturally or linearly to introduce an addendum to a part before feasibility analysis.

Remove Tool

The Flange tool has been renamed to the Remove tool. You can either remove or partition a selected surface from the part.

New Unflange Tool

Open linear bends as well as flange surfaces on user-defined reference surfaces.

Trim Angle Analysis

Analyze the trim angle of the free edges of the part with reference to the draw direction using the Show/Hide Trim Angle context menu.

Part Edge Analysis

Analyze part edge movements during forming using the Tracer tool. The initial point of contact of the selected edge is shown along with the final point and a measure of minimum and maximum distance between them on the part surface.

Tryout

Forming Using Solids

Perform a single or multistage forming simulation with sheet metal modeled as solid elements like shells for a single blank only. The initial blank definition needs to be solid geometry. The materials in the database are updated to be compatible with solids. The workflows for model setup, analysis, and results visualization with shells are the same as for sheets. An FLD (Forming Limit Diagram) based on formability is also available for solid top and bottom surfaces.

Tool Creation By Offset

Create tools by offsetting with a value smaller than the sheet thickness. An offset value no smaller than a quarter of the sheet thickness is recommended. This is available only for forming shell elements.

Auto Gravity Model Setup

Quickly add a gravity operation and automatically set up the tools before a forming operation using the Add Gravity option in the operation context menu. This will automatically copy bottom tools and pins from the forming operation to the gravity operation to reduce setup effort.

Trim Operation

You can now select the surface - Top, Middle, or Bottom - from which the trim/pierce/lance line is extracted to improve trimming robustness. Geometry edges can now be selected for trim/pierce/lance objects.

Trim Angle Analysis

Analyze the trim angle of the free edges of the part with reference to the draw direction using the Show/Hide Trim Angle in the context menu.

Part Replace

Replace any tool or blank, except for drawbeads, with a part in the model or from a file. The replaced part is deleted from the session.

Tool Mesh Parameter Option

New options to select **Individual tools** or **All tools** for the tool mesh parameter have been added to the Preferences. This setting determines the mesh parameters for the tool meshing. Individual tools were used in past releases, and this is the default.

Predict

Select geometry edges for blank shape and trim line prediction.

Springback Compensation.

Export springback compensation data (point cloud and deformation vector) in a text file format supported by CATIA and UGNX. This enhancement allows data exchange with CAD software to perform geometry based springback compensation. The springback compensation distance contour sign can now be flipped with a new result type.

Edge Strain

Contour edge strain can now be applied on free edges of the blank to study potential failure due to edge fracture.

Auto Callouts

Automatically create callouts based on Max (upper limit) and/or Min (lower limit) value. One callout per iso contour zone is created.

Tracers – Draw-in

Select a reference step to measure draw-in more accurately. By default, the first step is used as reference, but now it can be changed to any step, preferably to the end of binder close for apt comparison with physical tryout results. The draw-in measure dynamically updates with the animation.

Additional Changes and Enhancements for Forming

- Material: Barlat yield criteria can now be selected for Yoshida hardening law.
- An edge or line body can now be selected to define the draw direction.
- Orientation angle and center output are now included in blank nesting reports.
- Trim angle contour is now automatically enabled at blank fit and nesting to facilitate the best orientation selection for lock rotation nesting mode.

New Features and Enhancements 2022.0.1

The Inspire Form 2022.0.1 release includes the following enhancements and resolved issues:

Enhancements

- Stroke definition for binder and bottom pad is now supported
- Self-contact for solid blanks is now supported

Resolved Issues

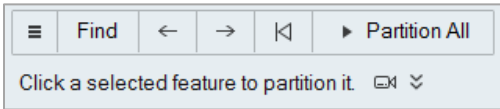
- Fixed an issue with dark patches in the report when published from dark mode
- Fixed an issue with prefixes in callouts
- Fixed an issue where frames were skipped when animated through all operations
- Fixed an application crash when automatically adding a gravity operation
- Fixed an issue with animation speed when tracers were on
- Fixed an application crash when loading models from Inspire Studio
- Fixed an issue ending runs from the Run Status window
- Fixed an issue where the run fails to start for non-planar solid blanks

Learn More About Inspire Form

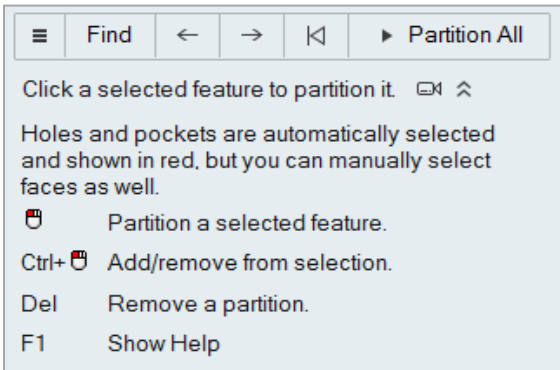
You can learn more about new and existing features in Inspire Form using the following resources:

In-Application User Assistance

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

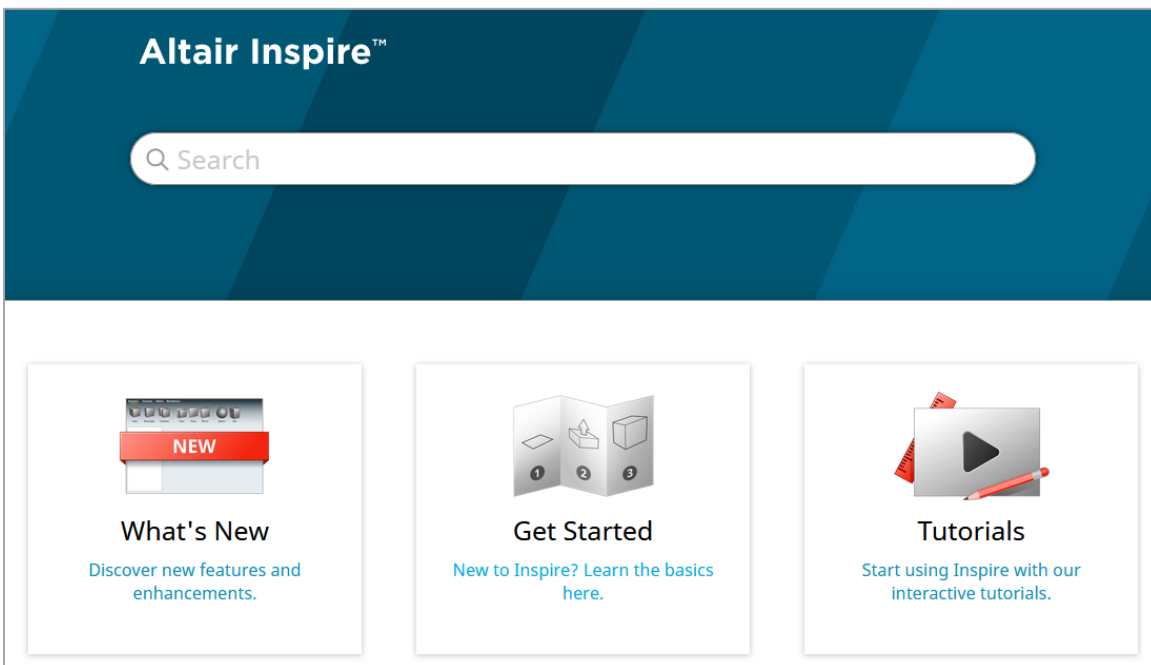


Click to view additional tips and shortcuts. Some tools also include a video .

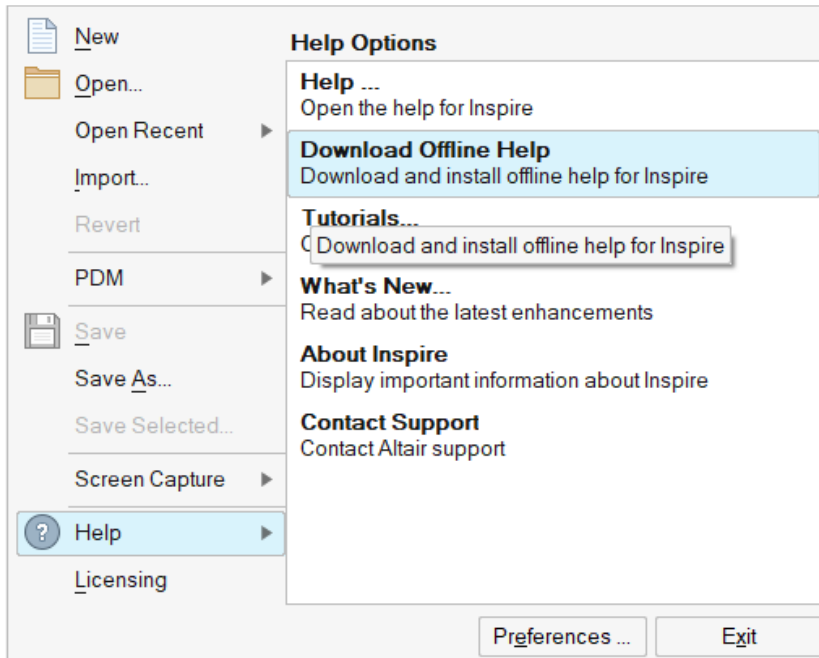


Online and Offline Help

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.



Supported Languages

The language for the user interface and online help can be changed in the Preferences under Workspace > Language. User interface text is available in English, Chinese, French, German, Italian, Japanese, Korean, Portuguese, and Spanish.

The online and offline help is available. If a language is selected in the Preferences that is supported for the user interface text but not for the help, the English help is shown. Similarly, if an unsupported language is selected in the Download Offline Help dialog, the English offline help will be downloaded instead.