ALTAIR
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RELEASE NOTES

Altair® Inspire™ 2022 and 2022.0.2

New Features and Enhancements 2022

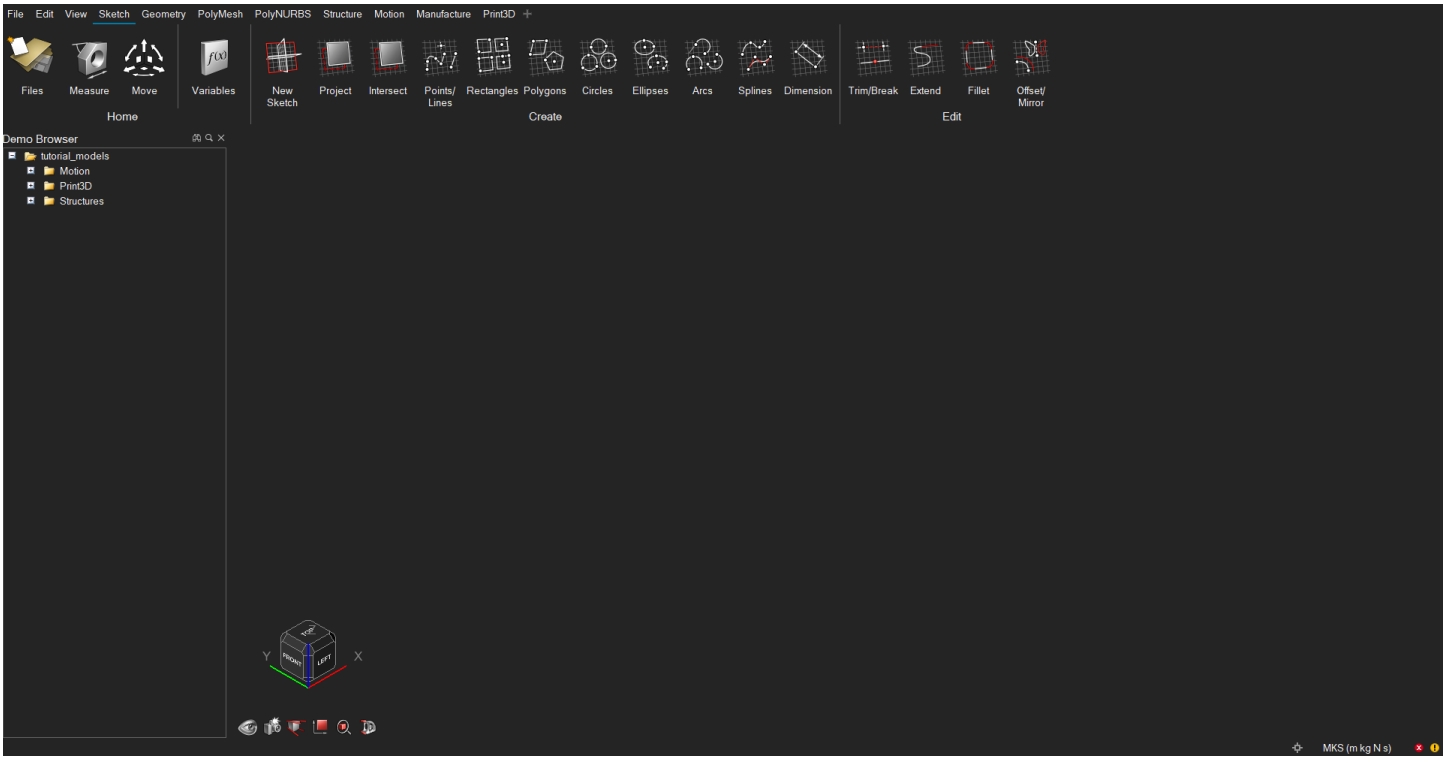
Release Highlights

- Dark Theme
- Design Explorer
- New Geometry and PolyNURBS tools
- Binder-Sintering Analysis for Print3D

General

Dark Theme

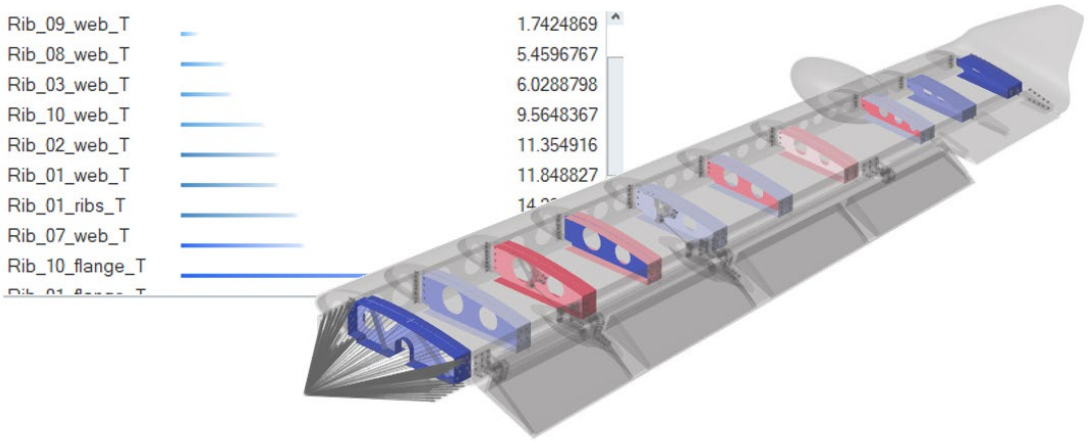
Dark Theme is now supported throughout Inspire. You can change the theme in the **Preferences** under **Workspace > Theme**.



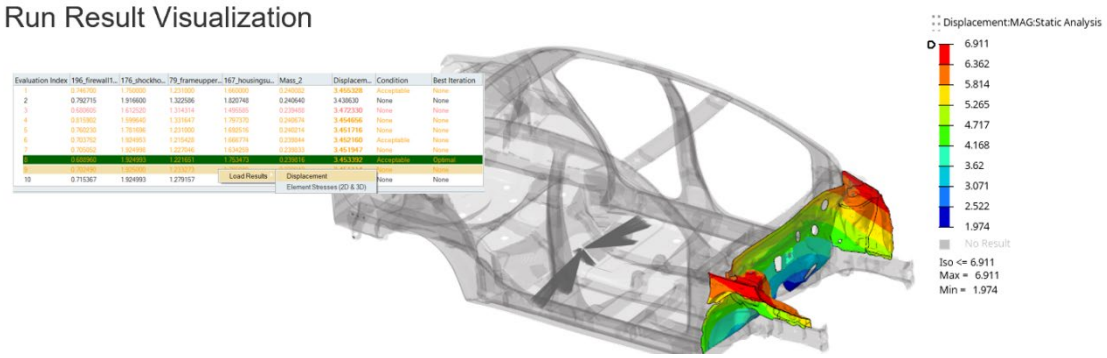
Design Explorer Ribbon

A new ribbon has been added with tools enabling you to explore, understand, and improve your system's designs using methods such as design-of-experiments (DOE) and optimization. By using Design Explorer, you can make better decisions and optimize the performance, reliability, and robustness of your systems.

Using DOE, you can find the Design Variable Sensitivity of design variables and measure trade offs.



Using Optimization, you can improve the system’s overall performance, stiffness, or stress.

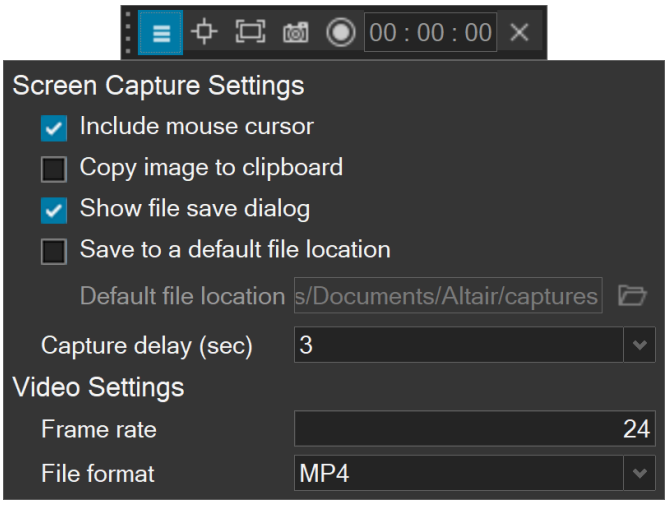


Python Debugger

A new Python debugger has been added for this release and is accessible from the Extension Manager.

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

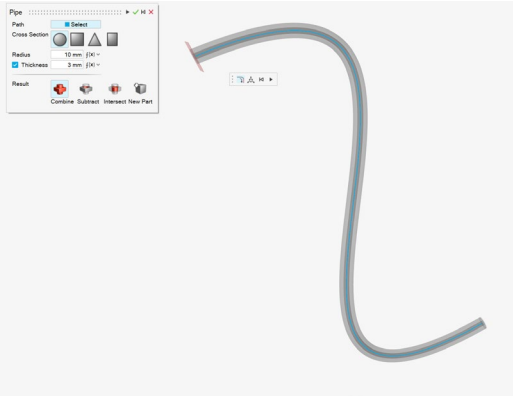
When outside of sketch editing mode, you can now press the **S** key and select a face to create a new sketch. Once in sketch mode, you can quickly access the most commonly used sketch tools with the following shortcut keys:

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.

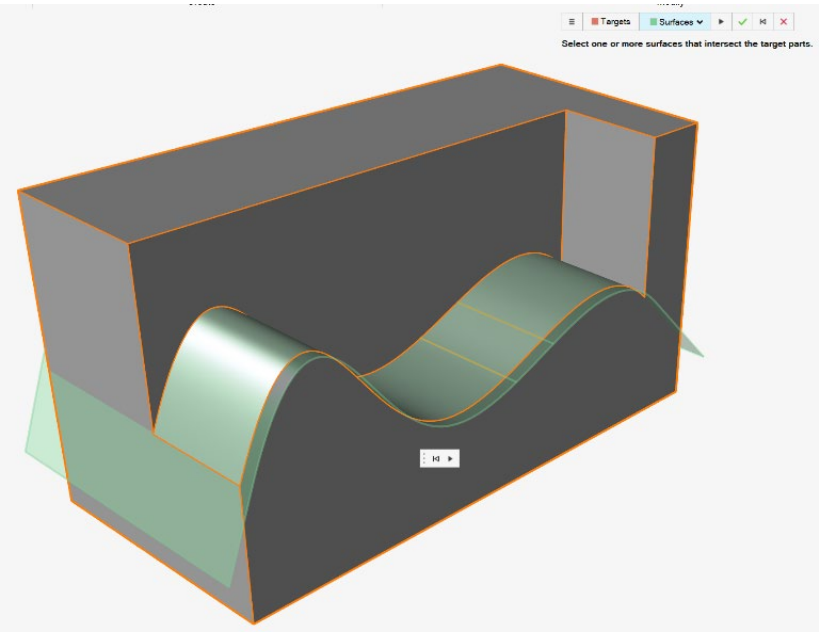
New Pipe Tool

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.



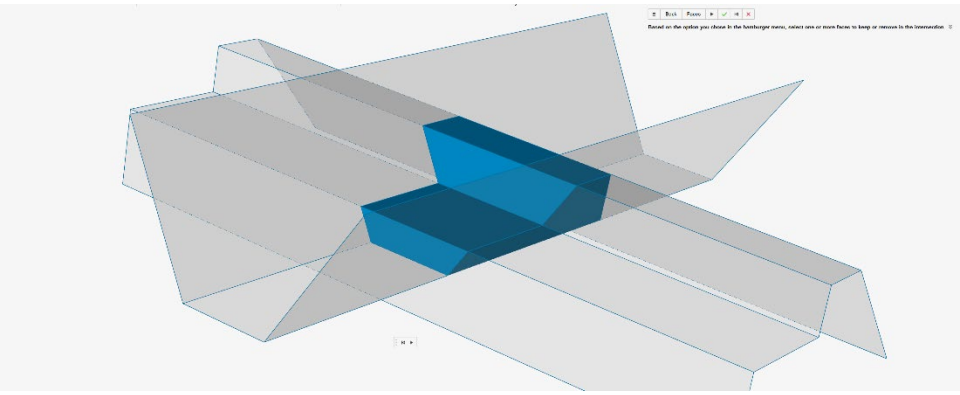
New Slice Tool

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.



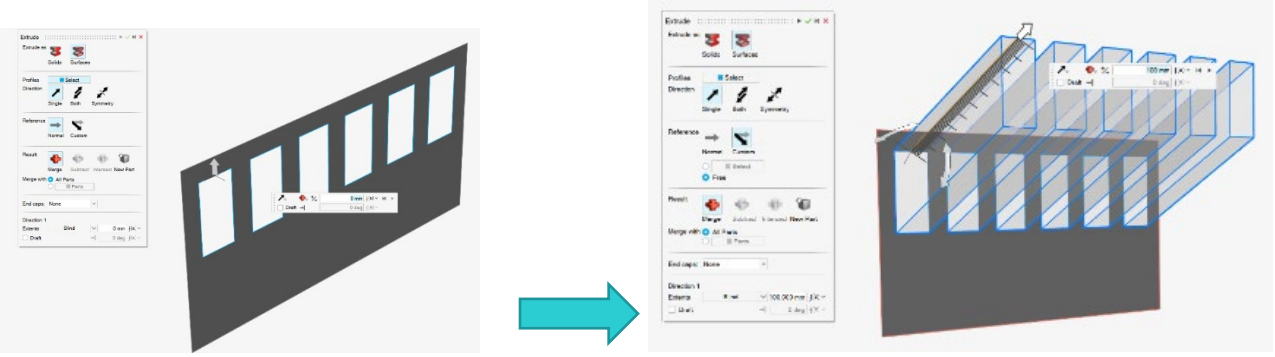
New Intersect Surfaces Tool

You can now retain only the intersecting portions of one or more parts.



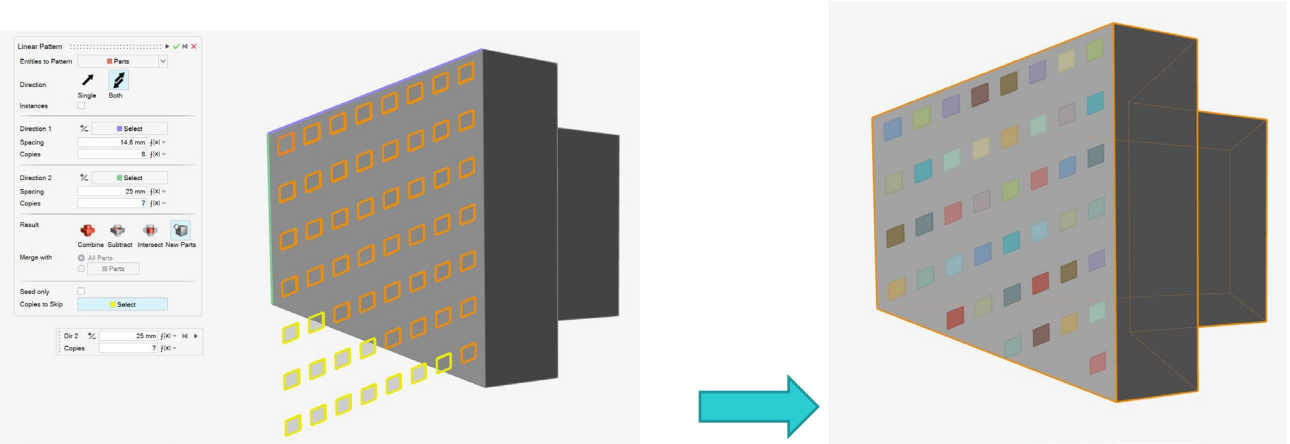
Extrude Tool Updates

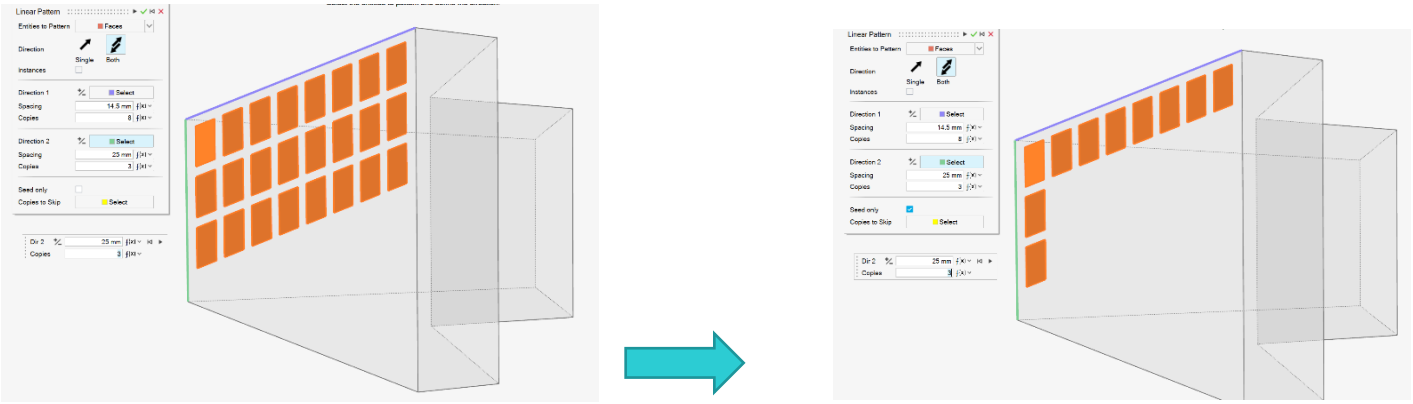
The Extrude tool has been enhanced for this release. You can now extrude all types of profiles, as well as 2D surfaces in one or two directions, to create solids or 3D surfaces with optional end caps. In addition, you can now extrude in a custom direction by either (1) selecting a reference geometry 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

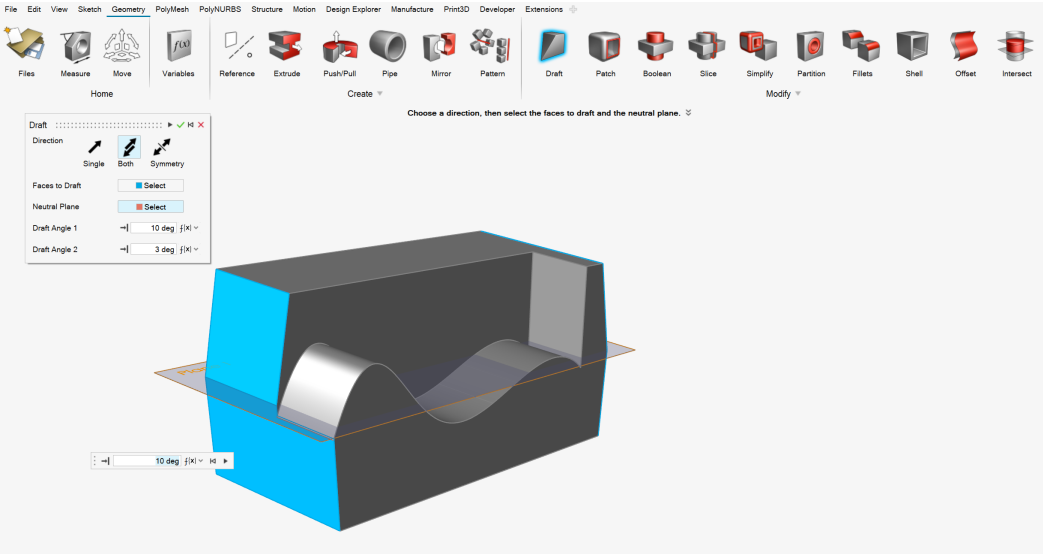
The Pattern tool has been enhanced and now allows you to perform Boolean operations on the results, as well as merge with all or selected parts. In addition, the new Copies to Skip option allows you to select copies to exclude from the pattern. For linear patterns, selecting the Seed Only checkbox will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



CAD Cleanup on Import

A new Run Import Diagnostics option has been added to the right-click context menu for CAD import construction features when the preference is enabled. To enable, go to **Preferences** and select **Inspire > Geometry > Import from CAD File > Fastest import > With diagnostics**.

Additional Changes and Enhancements for Geometry

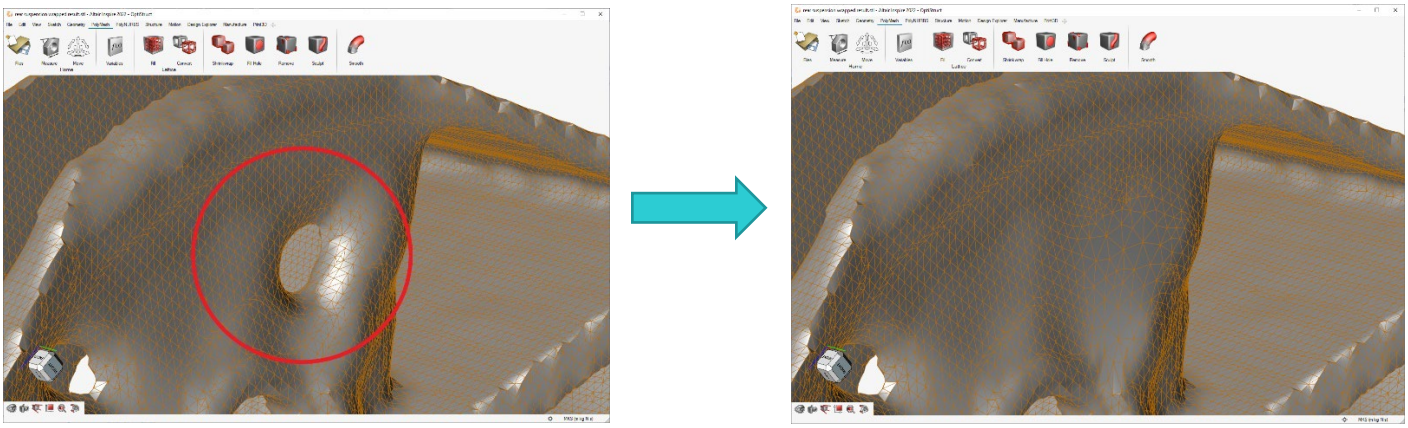
- For sketching, added a preference to toggle shading of closed regions on or off.
- The workflow for the Boolean Combine tool has been updated.
- Performance improvements for import of CAD files.
- Inspire now supports the following file format versions for import:

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

PolyNURBS

New Fill Holes Tool

The new Fill Holes tool on the PolyMesh ribbon allows you to fill holes in mesh parts.

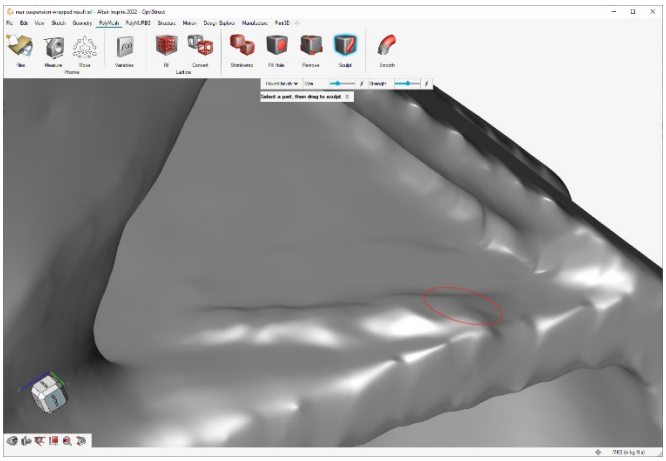


New Remove Tool

The new Remove tool on the PolyMesh ribbon allows you to automatically find small, disconnected regions of a mesh part and delete them.

New Sculpt Tool

The new Sculpt tool on the PolyMesh ribbon allows you to modify the original mesh using one of several brush tools.



New PolyNURBS Shell Tool

The new Shell tool on the PolyNURBS ribbon allows you to remove the cage face and create thin walls to generate a shelled PolyNURBS part.

Symmetry for PolyNURBS Fit

A new symmetry option has been added to the PolyNURBS Fit tool.

Structures

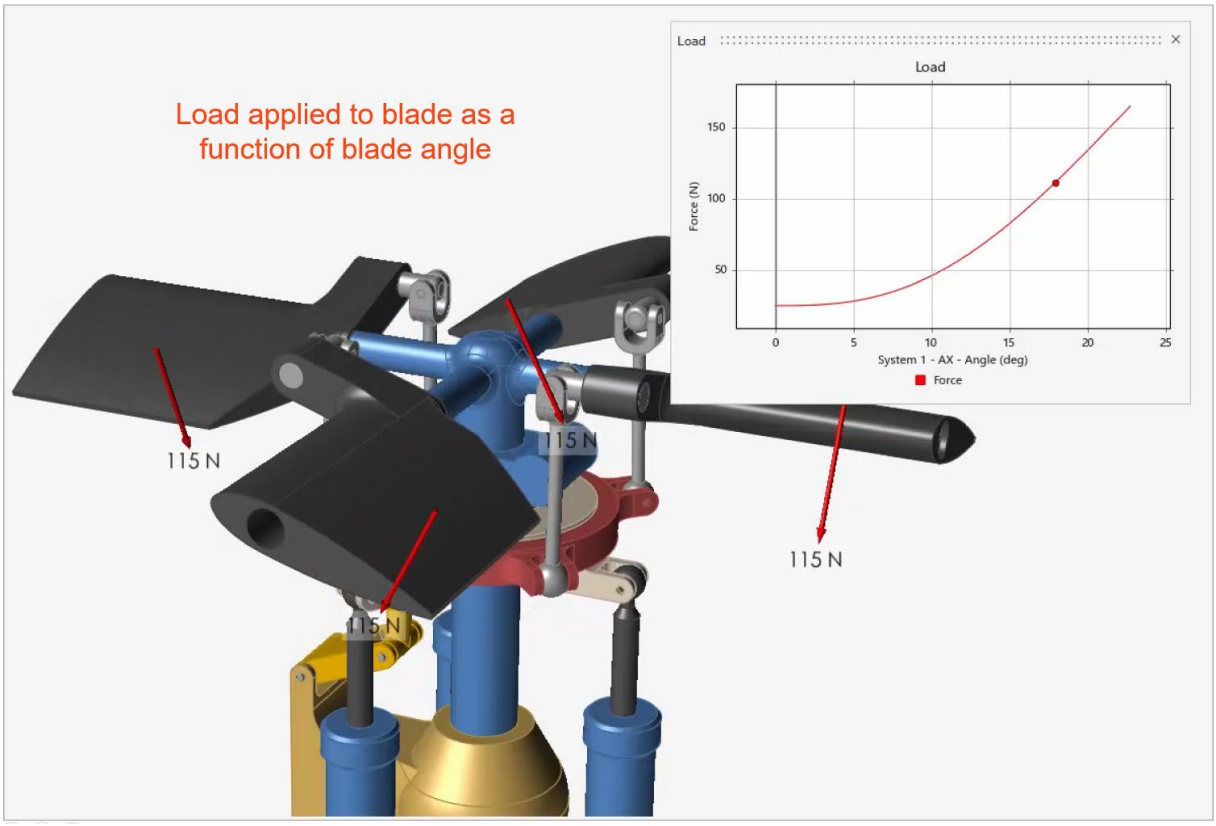
Spot Welds Tool Updates

You can now import and export spot welds in .csv format.

Motion

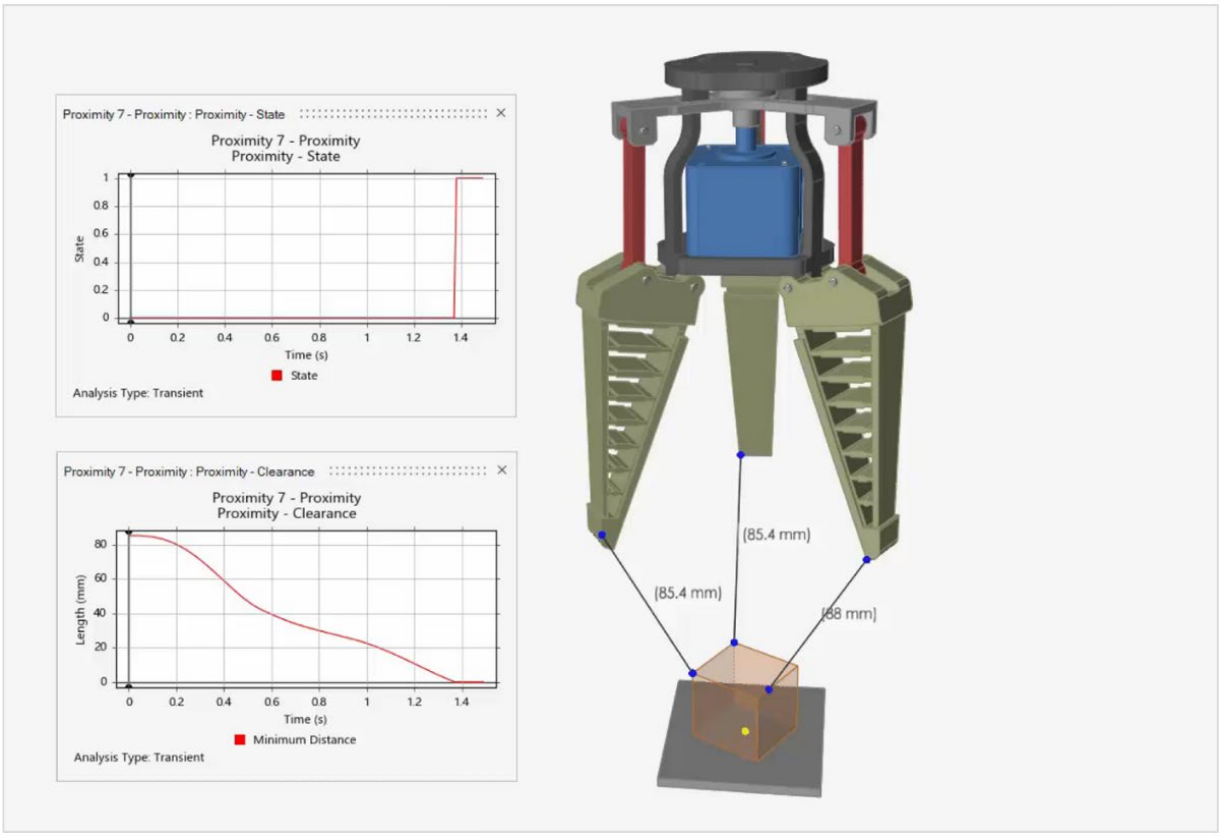
State Dependent Inputs

Users can now assign non-time dependent inputs to motors and actuators that are dependent on the state of motors, actuators, systems, or linear or angular measures. For example, torque can be applied to a motor that is a function of speed, or a force can be applied to an actuator that is a function of angular displacement.



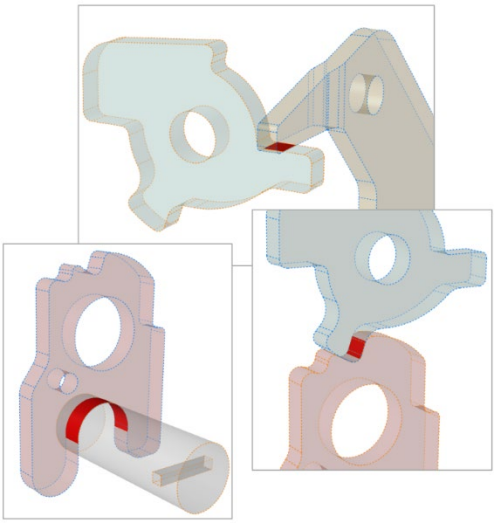
Proximity

A new Proximity tool has been added, allowing you to monitor the proximity between two parts before or during a simulation to detect when contact or interference may occur. End points are color-coded according to the part proximity (finite distance, touching, interfering). You can plot the “State” to see points in time of interference.



Find Initial Intersections

The Motion Contacts guide bar now includes an option to check for, and step through, initial intersections (overlaps) between contacting parts, which may present difficulties in solving the model. Intersecting regions between parts are highlighted in red, indicating where geometry corrections may need to be made.



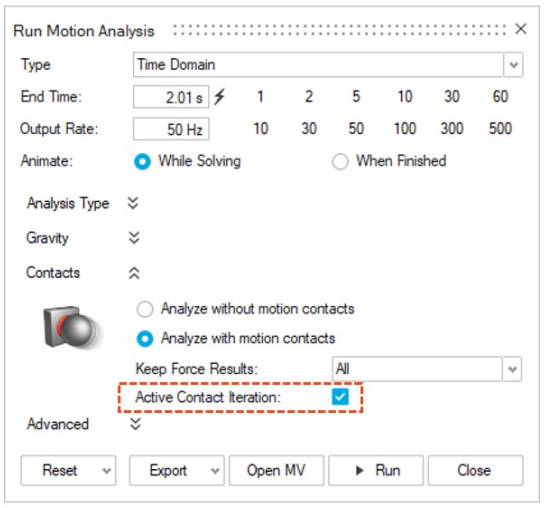
Active Contact Iteration (ACI)

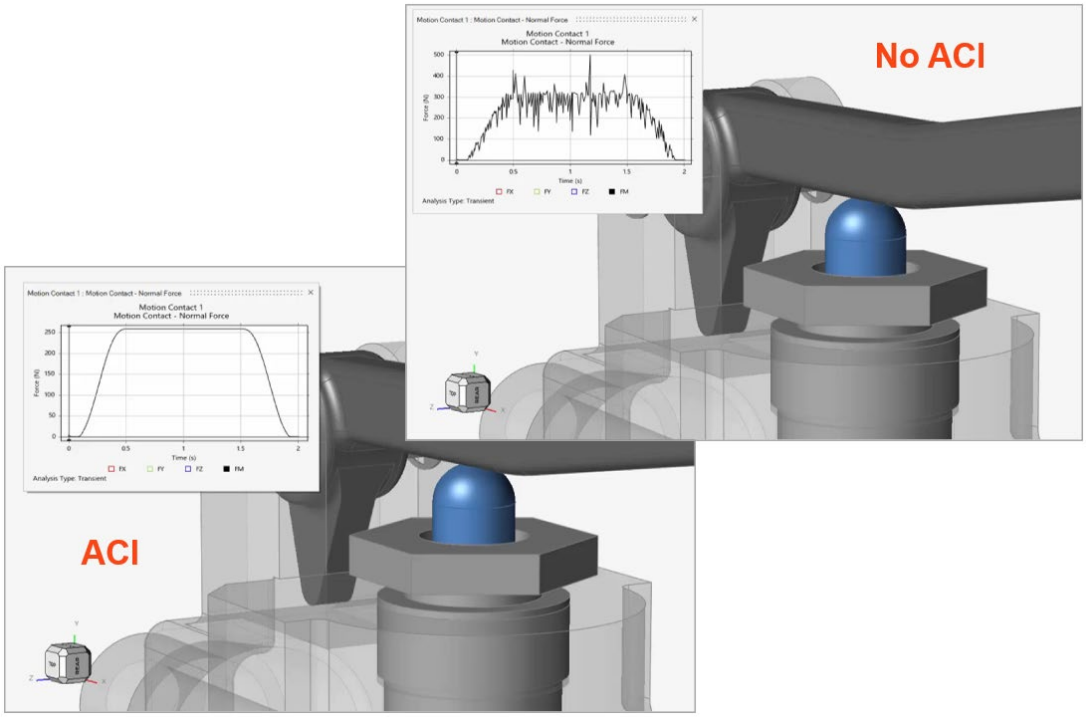
In certain models that involve contact between parts, the quality of the force results can depend on the frequency of updating of the matrix of partial derivatives of the solution (Jacobian matrix). In models where there may be a large time step or a high contact stiffness

relative to contact damping, the contact forces may appear noisy. ACI performs additional refinement to the solution for contact events, helping to reduce noise due to contact forces and provide more accurate force results overall.

Depending on the application type, ACI may improve speed or accuracy or both. You may obtain more accurate results and experience shorter run times in certain contact models. However, in some models, using ACI may also increase the simulation time slightly.

By default, ACI is turned on in models created in version 2022 and later, and it is turned off in models created earlier than version 2022.



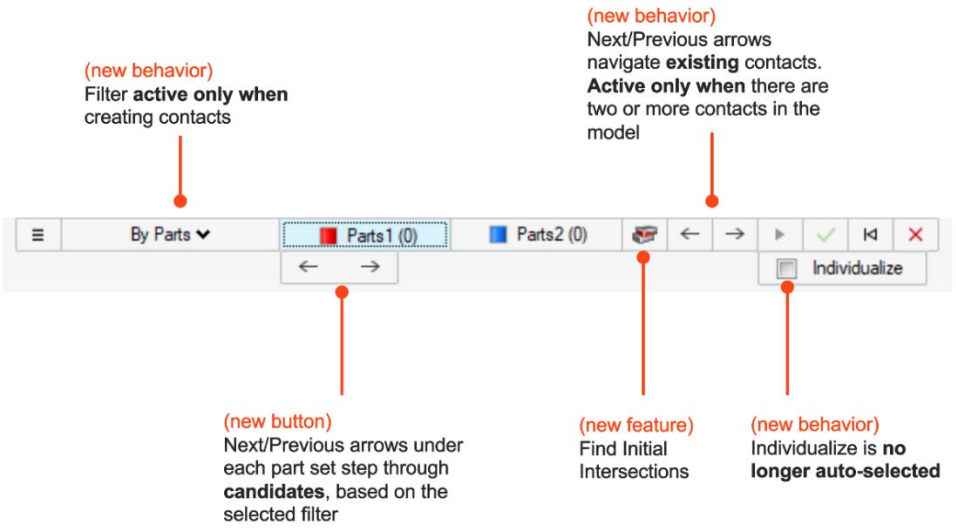


Additional Changes and Enhancements for Motion

Contacts Guide Bar

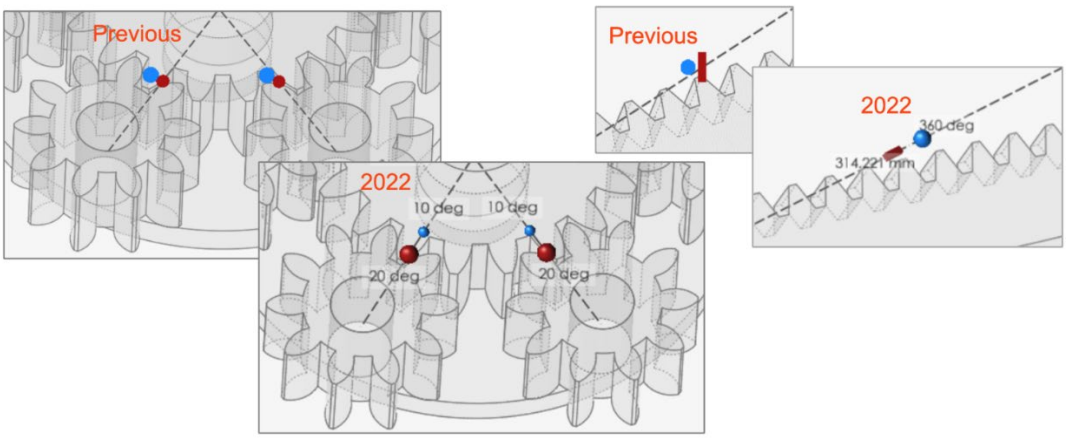
The workflow on the Contacts guide bar has been improved and clarified:

- The filter is active only when creating new contacts.
- Part sets now have arrows that navigate by the selected filter and place parts into the sets.
- The **Find Initial Intersections** feature has been added (see above).
- The navigation arrows on the main guide bar are now only for browsing existing contacts.
- **Individualize** is no longer turned on by default.
- Improvements regarding when buttons are active. For example, as soon as you click on an existing contact, the navigation arrows are activated and the filter is deactivated.
- There is one less mouse click when defining several contacts consecutively (you don't have to click in space).



Coupler Display

Couplers are represented by red and blue dots that are now along the line of sight of the couplers (as opposed to right next to each other). In addition, the scale value of the coupler ratio is now displayed next to the corresponding coupler.



Improvements to Flexible Body Handling

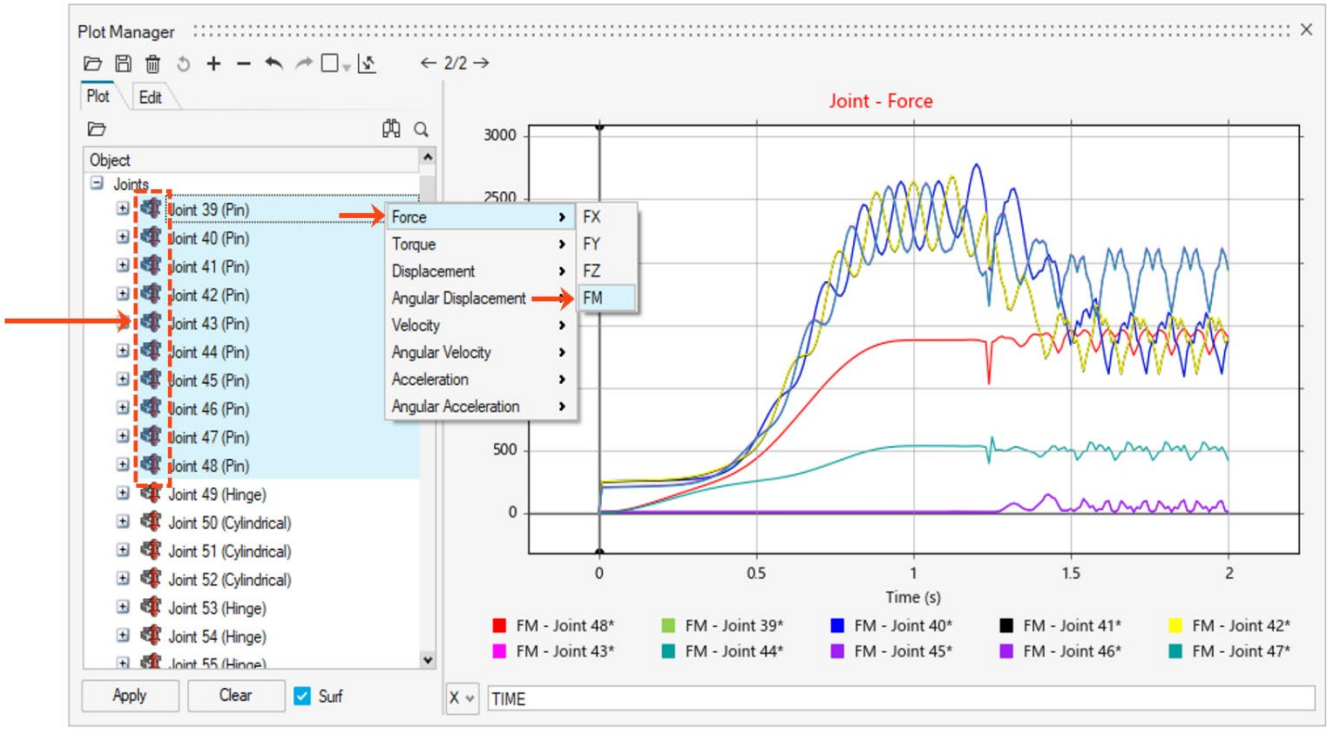
- The flexible body file (CMS .H3D) is now cached. Once a flexible body is created, it is automatically re-used if no changes are made to it or any of its boundary conditions.

- In addition to the option to specify # of modes, a Maximum Frequency cutoff option has been added.
- Loading of flexible body simulation results is now ~2 ½ to 3 ½ times faster than previously.
- When a part is made flexible, the part icon in the browser changes from the default rigid (cube) representation to a flexible part representation.

Plot Manager

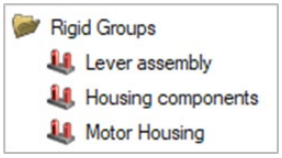
Users have more options for accessing output results. There are now also fewer clicks involved to get desired outputs.

- If there are multiple objects of the same class in the model (ex: joints), you can now multi-select and plot the same result for several components at once.
- If there is an empty plot inside the Plot Manager, you can click on an object on the model (outside of Plot Manager) to plot results for that object.
- If there is a callout plot actively displayed on the screen and you enter the Plot Manager, the callout plot populates the first page of the Plot Manager.



Rigid Group Browser Display

Rigid groups are now indicated by a new icon in the Model Browser.



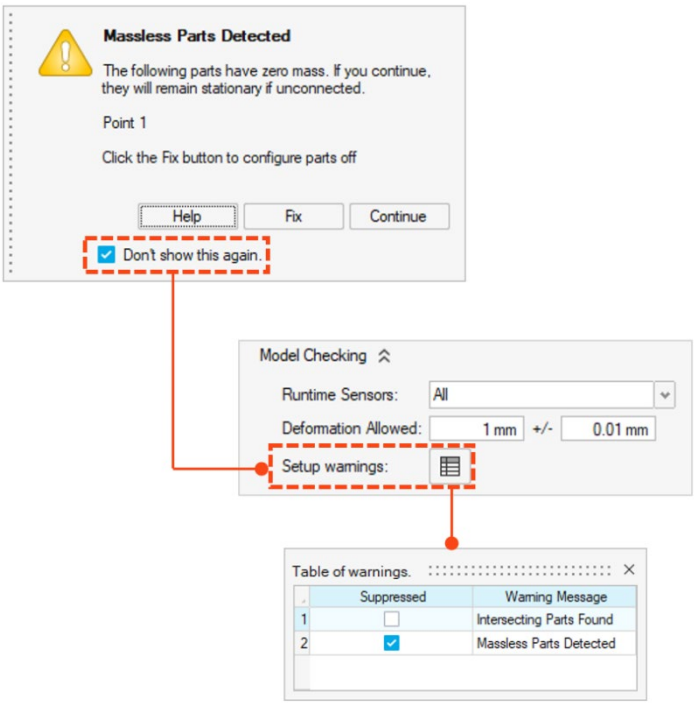
New Output Results for Systems and Measures

With the introduction of state-dependent inputs, outputs from systems and measures can be used as a guide in defining the independent variable range for the input.

- System Displacements are now plotted in relative coordinates, with the initial state beginning at 0. Previously, they were presented in Global position coordinates.
- Systems have a new Rotations Cyclic output (w/ angles in Ax, Ay and Az) (rotations alternate between 0^0 and 180^0).
- Systems also have a new Rotations Total output (w/ angles in Ax, Ay and Az) (rotations greater than 360^0 are summed).
- A new Travel output is available for measures, where the relative starting value is always 0.

Setup Warning Messages

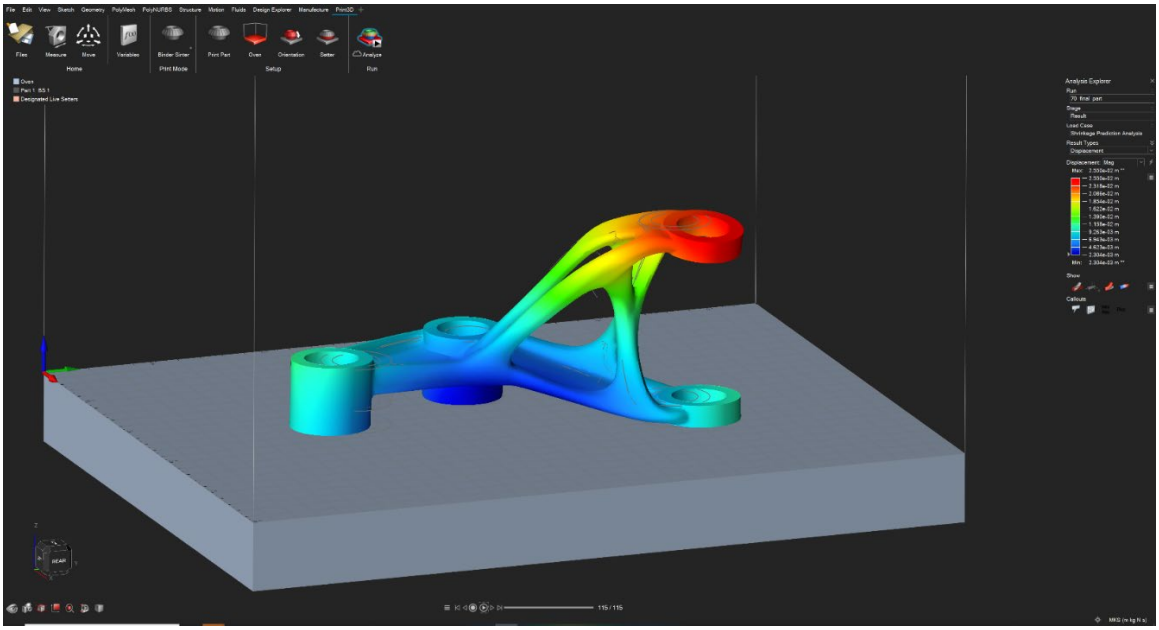
Warning messages that are displayed at the time of solve can now be turned off, so they do not display again in the same session unless the user reactivates the message. A **Setup warnings** feature located under Model Checking in the Run Settings dialog allows the messages to be unsuppressed and reactivated as desired.



Print3D – Binder Sinter

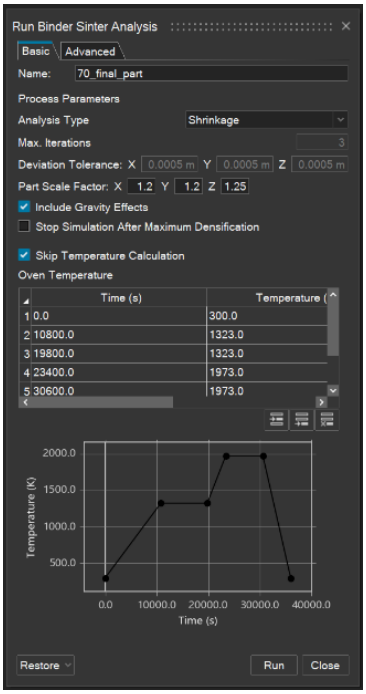
New Binder-Sinter Print Process

A new 3D printing process has been added for this release. Inspire now allows you to define the print part, oven, orientation, and setters for a binder-sinter simulation, and then perform a shrinkage + compensation analysis.



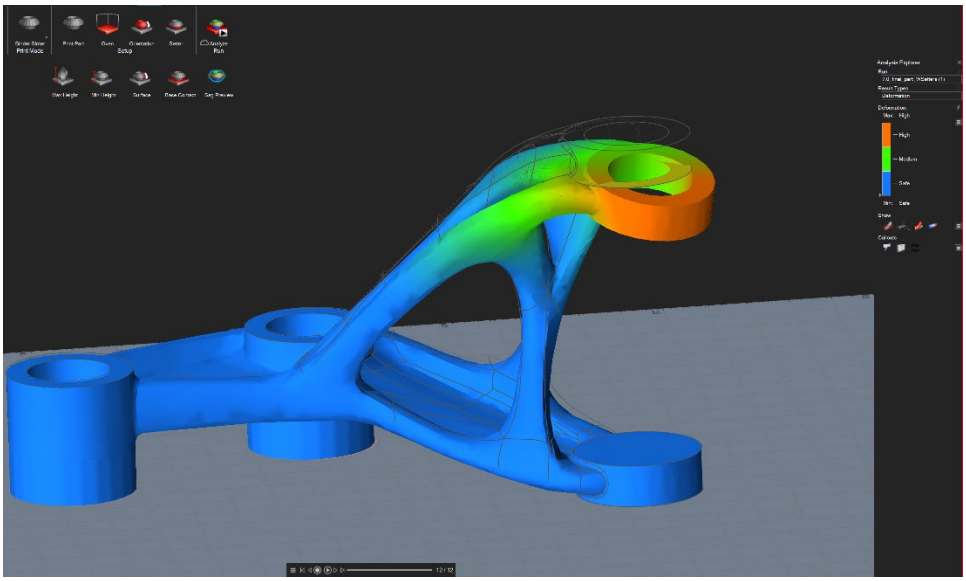
Quick Setup for Binder-Sintering

Just select the printed part, define the setters, and configure your oven curve. Inspire Print 3D can predict the green part based on the shrinkage compensation.



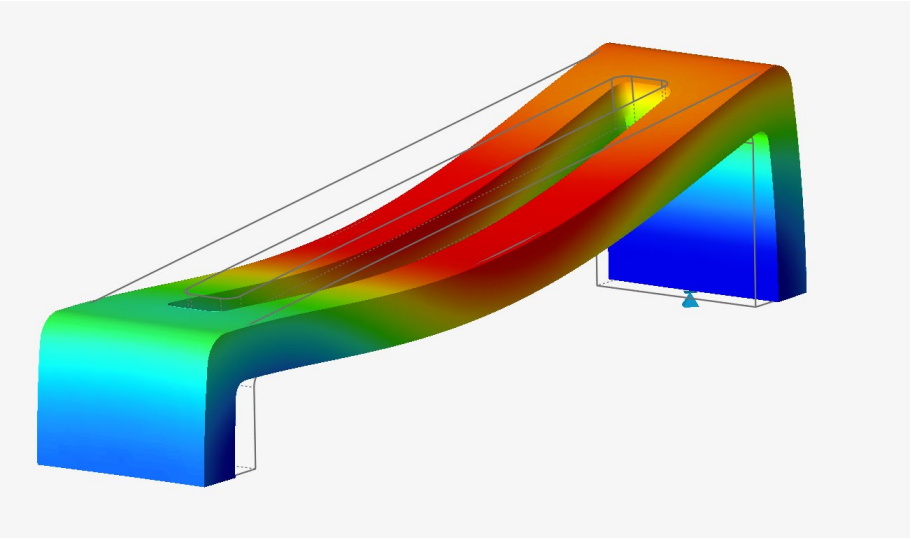
Sag Prediction

The Sag Preview tool shows what the expected deformation of the part will be after the sintering process. The preview appears in seconds and can be used to define setters or improve the part orientation.



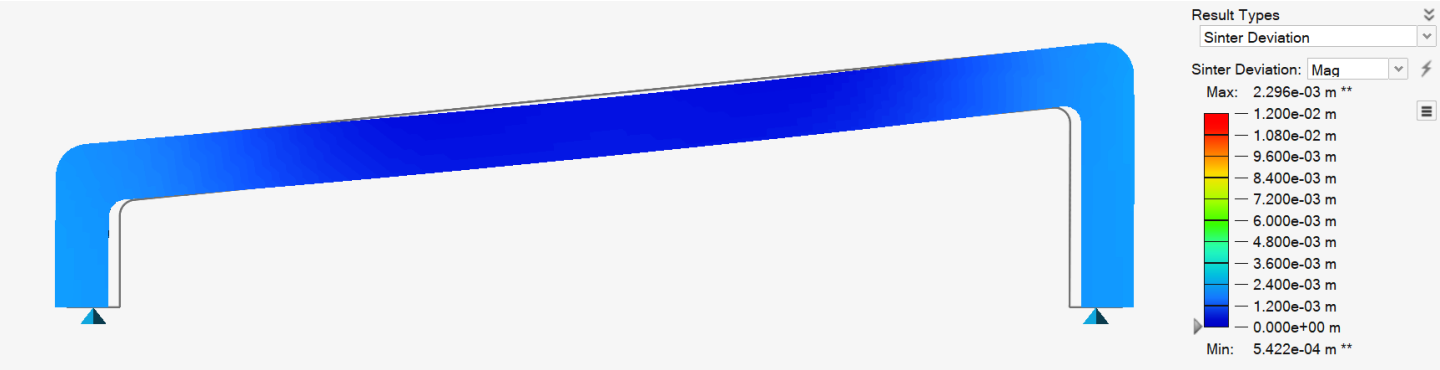
Results Deformation

You'll also be able to measure the maximum deformation of the part during the sintering process, so you can make decisions regarding setters, orientation, and process conditions.



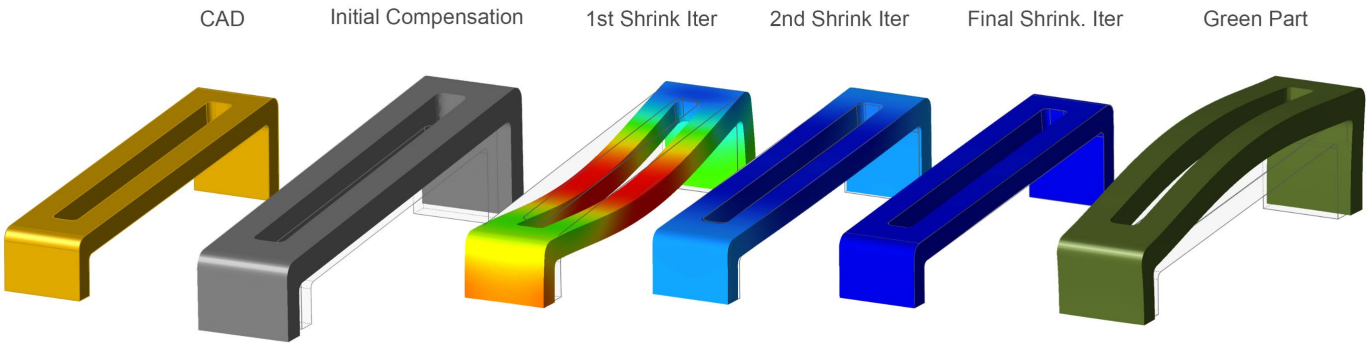
Sinter Deviation

Run a sintering analysis and compare the deviation from the original CAD shape you get after sintering.



Export the Green Parts

Finally, you'll be able to export the green part (compensated shape) to be printed and sintered.

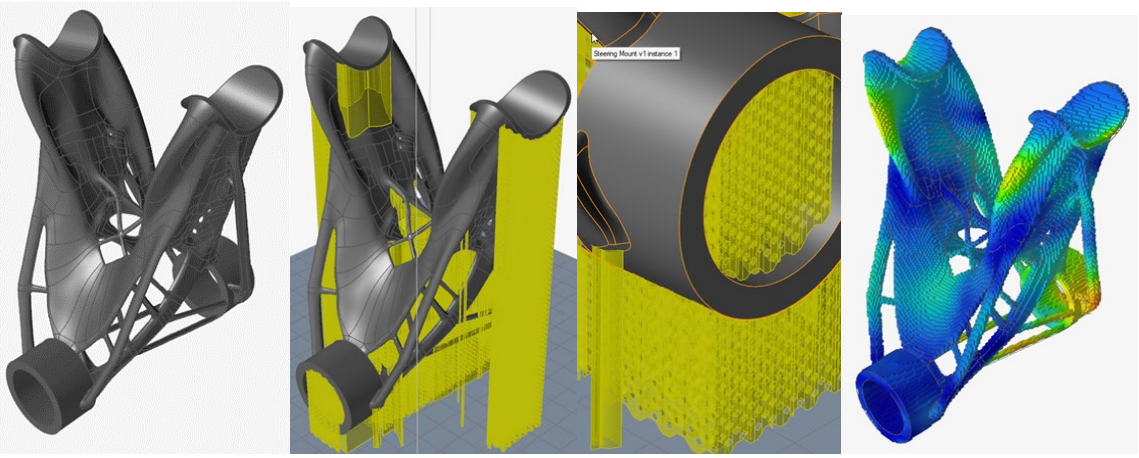


Print3D – Selective Laser Melting

Inherent Strains Technology for SLM printers

A **new 3D printing solver** based on the **Inherent Strain** approach has been added for this release.

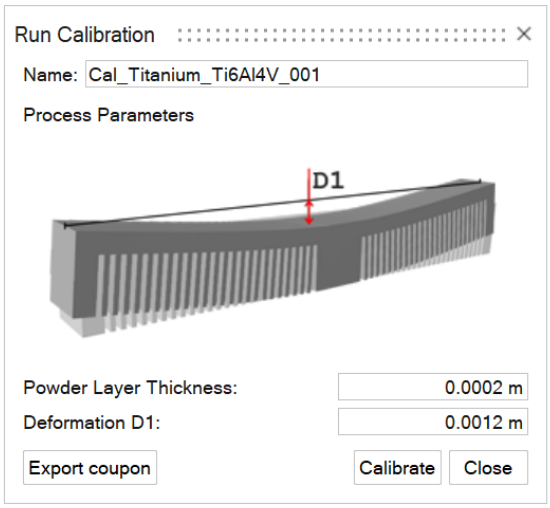
The inherent strain technology computes the deformation and residual stresses for a given model. The advantage of this approach is that it uses a calibration process to determine solidification shrinkage for a given printer and printing setting without the need for thermomechanical properties of the material. This solver only computes a mechanical solution after the calibration, so it is considerably faster than the full thermomechanical one for calculating the final deformation.



Calibration

The inherent strain solver needs a calibrated inherent strain as its input. Inspire Print3D now includes a calibration tool that allows you to easily calibrate the material before running the print simulation. You can also store different calibrated inherent strains for different layer thicknesses or other printing conditions and apply them easily when running simulations.

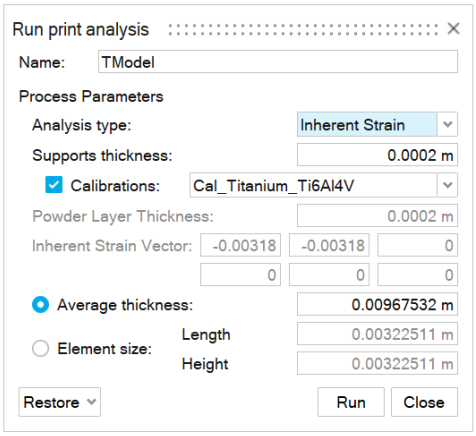
Calibration



Inherent Strains Calculation

To use the inherent strain solution, a new analysis type has been added to the Run Print Analysis window. The inherent strain analysis runs the simulation by specifying the inherent strain either manually or from the list of calibrations.

The inherent strain solver computes the part deformation and stresses generated during 3D printing and after removing the supports.



Resolved Issues

- You can now create spot welds by importing them via a .csv file. [IN-26750]
- Columns are now saved/restored in browsers upon relaunching Inspire. [IN-19375]

Known Issues

- Undo is not supported in the Design Explorer.
- Crash after rotating a model with existing SimSolid analysis results that contain fastener reaction forces. [IN-26783]
- Assembly instances can cause multiple analysis results to stack on top of each other. [IN-26846]
- On some surface models, pressures are analyzed using the wrong direction. [IN-26590]
- Hover highlighting is incorrect in the Results Explorer when Show/Hide Initial Shape is turned on. [IN-24980]
- Wrong Radial Bead Pattern causing an optimization error. [IN-16311]
- If you run a motion analysis and then attempt to perform an Analyze Part within the motion context, and the part has no resulting loads associated with it, the analysis will fail partway through the solution process with no warning message

indicating why it failed. To resolve, verify that the part being analyzed has other loads besides only gravity associated with it. [IM-4157] [IM-4269]

- Depending on your install configuration, the Inspire run history path, located under Preferences, may point to your Documents folder linked to Microsoft OneDrive. In this case, the flexible body creation process will fail and warn of a missing .H3D file. To resolve, change the run history path to a folder not located on Microsoft OneDrive.

New Features and Enhancements 2022.0.2

Resolved Issues

- For PolyMesh tools, the Remove Island API is now supported in batch mode. You can find small, disconnected regions of a meshed part and delete them in batch mode. [IN-27588]
- Fixed an issue where stamping and extrusion draw directions were not created. [IN-27790]
- Temperature load information is now allowed in the model dump. [IN-26286]
- For the Fit PolyNURBS tool, a check box to enable/disable symmetry plane was added. [IN-27614]
- Auto Find has been turned off by default. [IN-27623]
- Fixed an issue where Shift + box selection sometimes didn't deselect a PolyNURBS. [IN-26033]
- Fixed an issue where a failed Pattern operation wasn't indicated graphically. [IN-26082]
- Fixed an issue where, after creating a sketch, dragging the mouse didn't snap the sketch entity to the grid. [IN-27376]
- Fixed a failure to combine the parent part using the Pattern tool's Combine option. [IN-27660]
- Fixed a crash when using the Sculpt tool in the German user interface. [IN-27686]
- Fixed the % yield calculation from SimSolid results. [IN-27689]
- Fixed an issue where the desired optimization structure was not being captured when using the PolyNURBS Fit tool. [IN-27730]
- Fixed a construction history issue when entering and exiting Sketch mode. [IN-27797]
- Fixed an issue where Boolean input wasn't being processed when using the Extrude tool. [IN-27836]
- Fixed an issue where obsolete messages were not cleared from the Setup Warnings. [IM-4169]
- Fixed an issue where there was no F1 help link for the Proximity ribbon icon. [IM-4238]
- Fixed an issue where the FE solve would complete but there were no results if Analyze Part was performed on a part that contained no valid motion loads. Now, if there are no loads present on a part, and Analyze Part is requested, a warning message will appear, indicating there are no valid loads to send to the analysis. [IM-4157]
- Fixed an issue where an error occurred if a coil spring was defined by selecting a concentrated mass as the first part selection. [IM-4289]
- Fixed an issue where the tooltip for Proximity measure was not correctly translated in some languages. [IM-4226]
- Enhanced tooltips have now been added for all items under the Motion Run Settings dialog. [IM-4096]

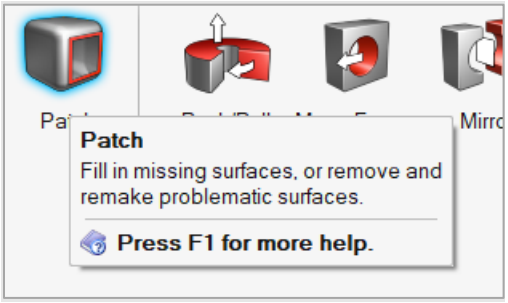
- Fixed an issue where a thermomechanical analysis would fail if run immediately after a calibration analysis. [INP-1600]
- Fixed a crash with Undo when using variables [IN-27921]
- Fixed an issue that allowed you to select faces from different parts when using the Pattern tool [IN-27996]
- Fixed an issue that imported thin solid parts in CATPart files as nonmanifold geometry [IN-27967]
- Renamed “Radius” to “Diameter” in the Pipe tool [IN-28000]

Learn More About Inspire

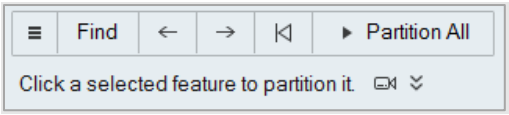
You can learn more about new and existing features in Inspire using the following resources:

In-Application User Assistance

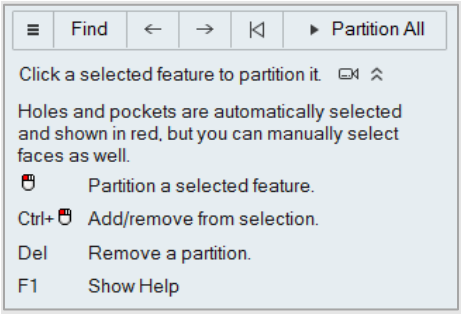
Inspire provides two types of user assistance. **Enhanced tooltips** appear when you hover over icons and other features. They describe what the tool does.



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

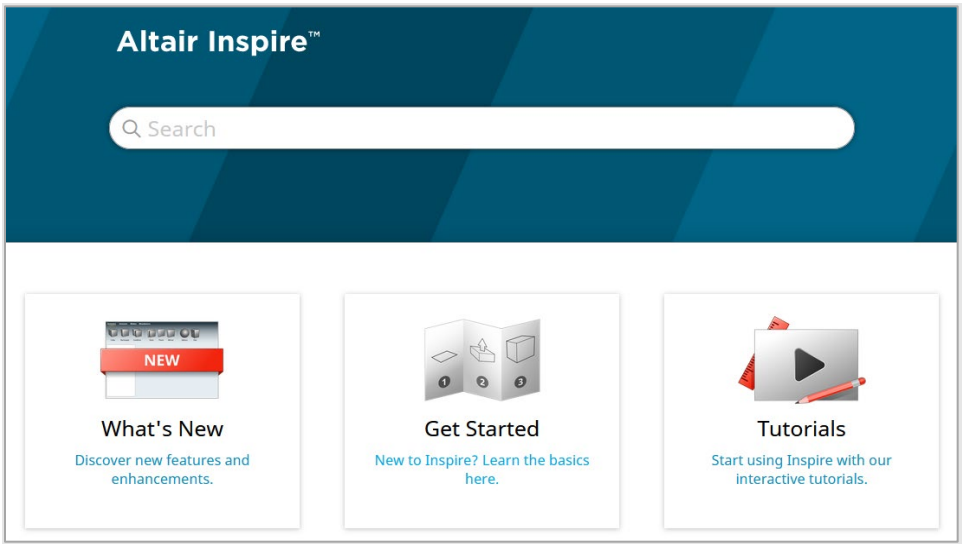


Click  to view additional tips and shortcuts. Some tools also include a video  .

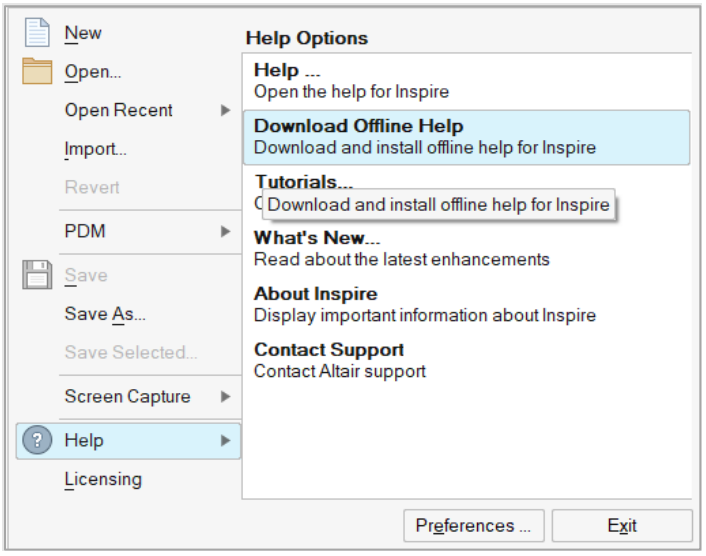


Online and Offline Help

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.



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 in English at the time of release, and in Chinese, Japanese, and Korean generally 1 to 2 months after release. 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.

