ALTAIR

ONLY FORWARD

Altair® HyperWorks® 2025

Release Notes

Updated: 01/09/2025

Contents

2025 Release Notes

1

This chapter covers the following:

Altair HyperMesh 2025 Release Notes

Aeroelasticity

No updates were made to Aeroelasticity in 2025.

API and Customization

New Features

- Python API Recording
  - On-demand recording of graphical user interface actions in the HyperMesh client.
  - Advanced entity selection capturing.
  - Parametrization of the generated code.

Enhancements

- Entity constructor now supports any attribute type as a keyword argument (previously limited to floats, integers, and strings).
- Enhanced Collection class instance methods:
  - New *filter* and *intersect* methods.
  - You can directly supply an entity object to the *contains* method.
- *CollectionByInteractiveSelection* supports defining a user message.
- Added new entity class method *getentitylist* to create an empty entity list.
- Tabular entity attributes are now supported.
- All attributes returning a list of numerical values now return a NumPy array when queried.
- Creation of HyperMesh Post Derived Load Cases (DLCs) is now supported.
- Added new functions providing interactive widgets:
  - *PlaneByInteractiveSelection* - returns a dictionary with plane base and normal coordinates.
  - *EntityListByInteractiveSelection* - returns a list of entity objects.
  - *EntityByInteractiveSelection* - returns a single entity object.
- New functions *hm_count*, *imprint_mesh*, *hm_getactiveplotcontrolmaxvalue*, *hm_getactiveplotcontrolminvalue*, *hm_getactiveplotcontrolvalues_bycollection*, and *hm_getactiveplotcontrolvalues_byentity*.

Known Issues

- Certain entity attributes are not yet accessible via the Python API. The coverage will be continuously improved over the coming releases.

Resolved Issues

- Executing scripts via **File > Load** or via drag-and-drop works correctly now.
- All entity subclasses are supported by functions populating a collection or a collectionset object.
- Passing and empty collection to *CollectionByAttached* caused a segmentation error. This issue has been resolved.
- When defining collections of load classes (*LoadForce*, *LoadPressure*, and so on), the specified entity class is now correctly respected.
- Collection objects contain the correct entities when constructed by using a list of entity IDs in a form of a NumPy array.
- The *plane_base* and *plane_normal* argument names have been corrected for all functions using a plane definition as an input argument.
- *CollectionByInteractiveSelection* was not working correctly for certain entity classes. This issue has been addressed.

Browsers

Enhancements

Updated Show in Browser Behavior

The Show in Browser context menu option now behaves differently for unnamed entities:

- If the entities selected on the graphics are within the defined limit, they will be sent directly to the browser without prompting the browser selector.
- If the selection exceeds the limit, the browser selector is displayed. This allows you to manage the selections.

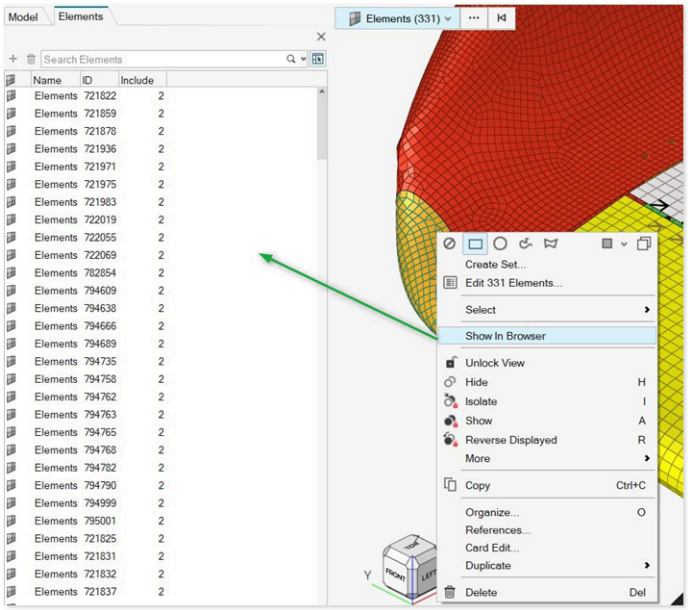

Figure 1:

Keyboard Shortcut for Show

From the white space of a browser, press the **A** key to use the shortcut for the Show context menu option.

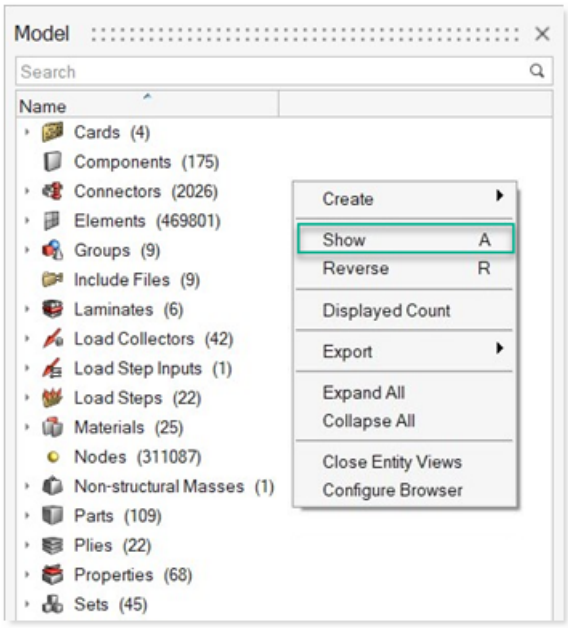

Figure 2:

Configurable Threshold Limit

The Preferences dialog now has an option to change the limit for the entity populations in browsers.

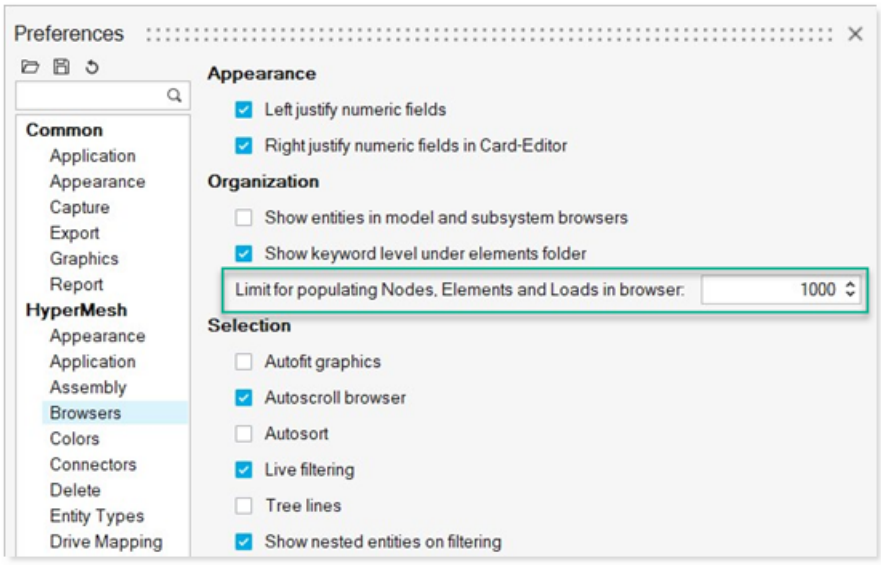

Figure 3:

Entity Defaults for Properties

Entity Defaults can now be configured for Properties within the Ansys interface.

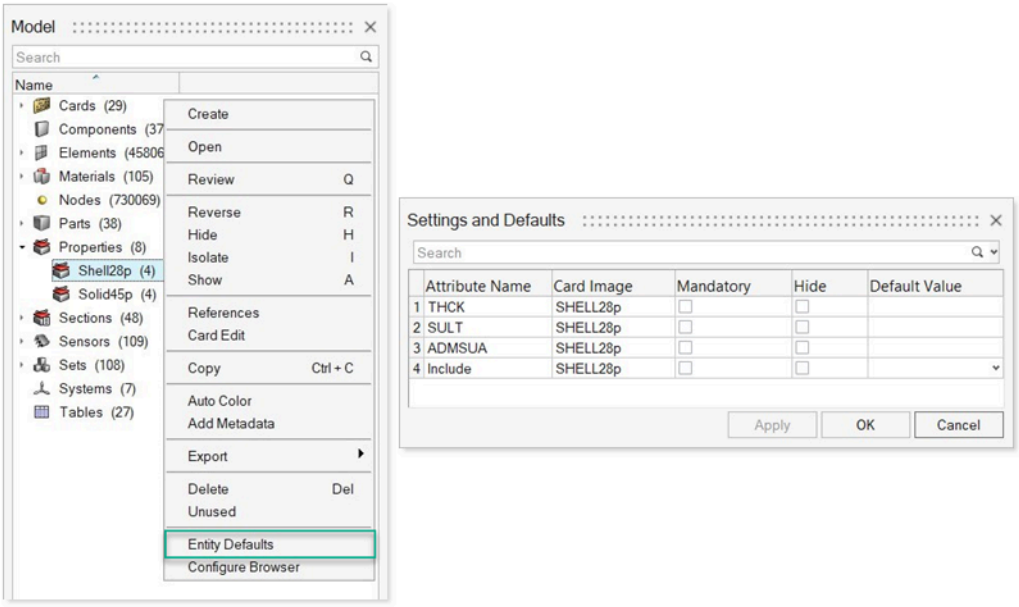

Figure 4:

Section Cuts from the Browser

From the browser, click the **entity icon** to activate section cuts.

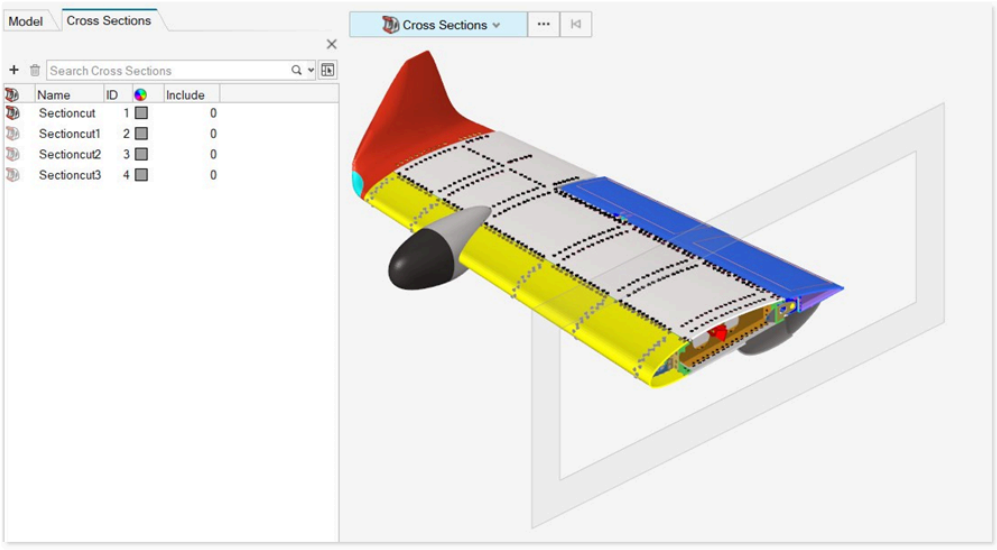

Figure 5:

Known Issues

The following known issues will be addressed in a future release as we continuously improve software performance:

- The Create menu keyword list is removed from the entity views upon loading a new model.

Resolved Issues

- Typing letters in the ID fields changed the ID to a random number.
- Deleting Solids or Connectors from the Model Browser now prompts you to delete the associated entities.
- The keyboard shortcut "R" was not working on the View entity.
- Nodal Coordinates were not updated in the Attribute columns upon moving nodes.
- Metadata added to the Feature entity was not shown in the Attribute columns.
- Sorting the Min/Max Occupied columns did not work when an empty value was present.
- Context menus did not match between the Entity Editor and the Attribute columns.
- When a component name contained curly braces, an application error occurred when setting the component as current.

Composites

New Features

Composite Stress Toolbox

Python API supports composite analyses covering the same feature set as the *compositeanalysis TCL command. Three variants exist as part of hm.Model(): compositeanalysis_byentity(), compositeanalysis_bycollection(), and compositeanalysis_byname().

Enhancements

Preprocessing

Offset visualization for Radioss (/STACK, /PROP/TYPE51) and LS-DYNA (*PART_COMPOSITE NLOC).
Performance improvement for spreadsheet I/O.
Composite Browser Ply/Views reorganized to pull-down icons.
*SHELL_GENERAL_SECTION support for composite absorb.

Composite Stress Toolbox

Numerical (table) results from the Composite Analysis Result Viewer can be copied to the clipboard with the **Ctrl** > **C** keyboard shortcut or via the context menu available in the result display area.
Performance has been improved for the analysis of large numbers of entities for engineering constants and load response.
Anisotropic thermal expansion coefficients are supported in various analyses for Multiscale Designer materials.

Known Issues

Preprocessing

- Comprehensive support of LS-DYNA material orientation review.
- Ply shape assignment error if multiple plies are selected if the new shape is the same as the shape of first ply in the selection.
- Ply icon shading in the Composite Browser as plies are shown and hidden.

Composite Stress Toolbox

- Load response contouring does not fully support symmetric laminates. While element contour already takes symmetry into account fully, the ply wise result selection is still lacking the symmetric layers.
- Contouring load response for PCOMPG relies on layer index instead of global ply ID.

Resolved Issues

Preprocessing

- A performance improvement has been made for laminate realization and detailed composite visualization.
- By node orientation method no longer fails if components are isolated.
- Ansys property absorb to ply based model - sets now generated and populated with elements, template properties with appropriate solver attributes generated.
- Loops of main and instance plies are no longer blocked.
- Catia Stackup file import no longer fails.
- Laminate export to spreadsheet no longer fails if one or more plies have no assigned material.

Injection Molding

- The Injection Molding Ribbon is now only available in the supported profiles OptiStruct and Abaqus.

Composite Stress Toolbox

- Tensor form shear strains are now used to correctly compute the principal strains (instead using of the engineering shear strains).
- Von Mises 3D failure calculation is no longer incorrect for use cases with specific relation of principal stresses.
- Drape tables are now correctly sized after re-realization to coarser mesh.
- Using *compositeanalysis with output to variable in a non-existing namespace will not create the corresponding namespace instead of producing an error.

Connectors

New Features

Auto Area Tools

This feature can be used to select Parts/Components and create Area Connectors within the a specified Tolerance. The result will mirror the result from the Auto Contact tool.

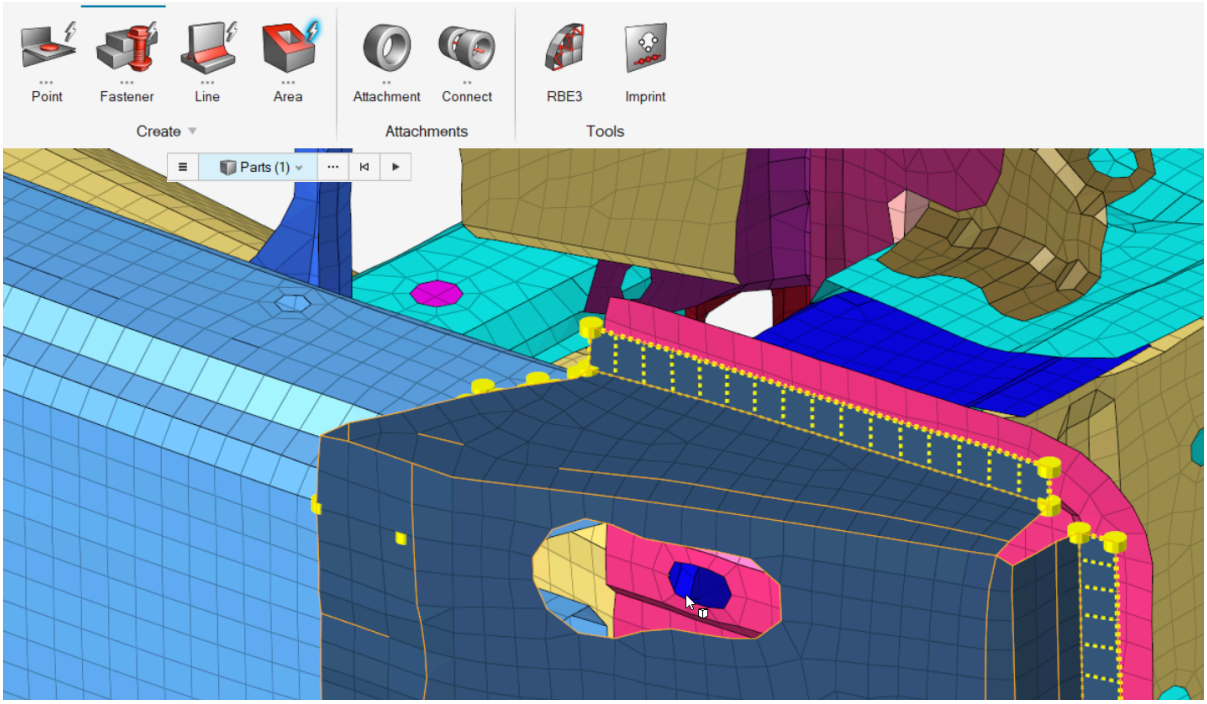

Figure 6:

Capture Shape Attachments

These new attachment options allow a shape selection and dimensioning. Any nodes within the selection on the part link is then selected. The different shapes are sphere, box, and cylinder.

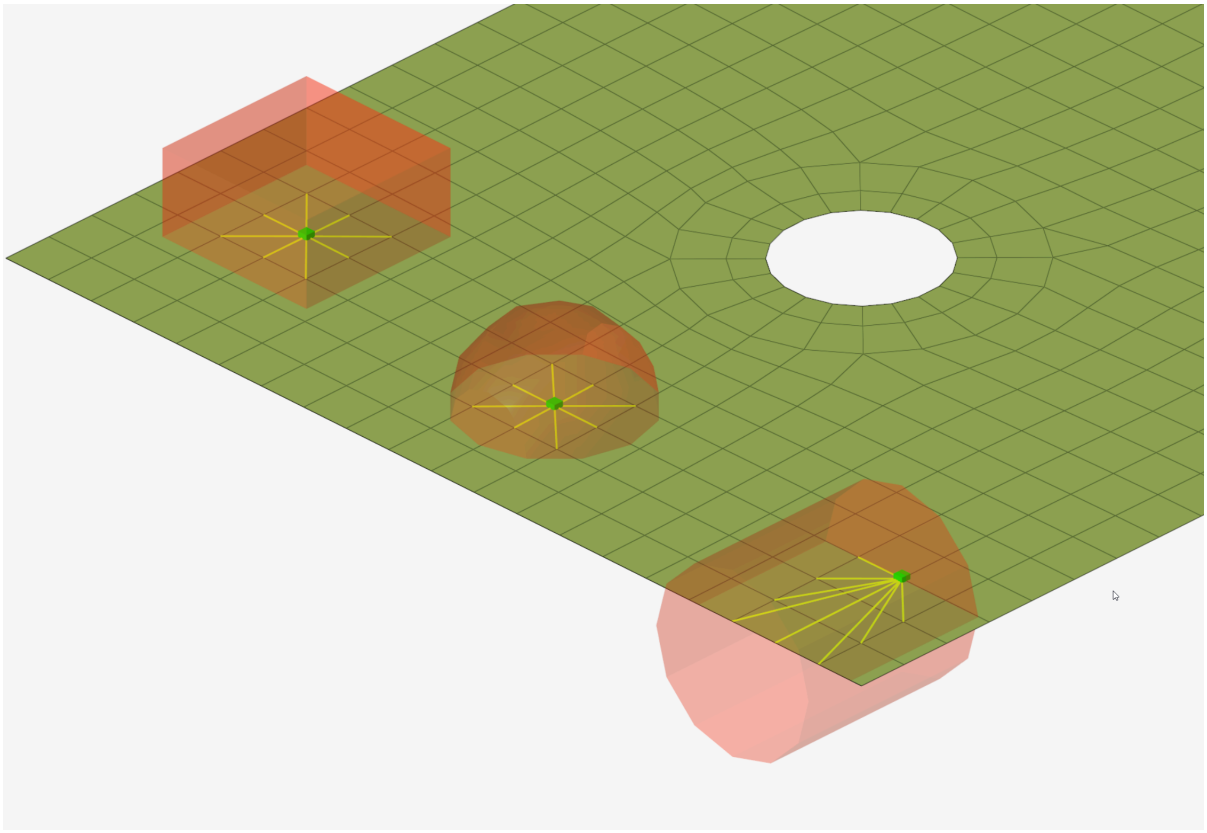

Figure 7:

Contact Connectors

This feature can be used to create contacts from the connector. This realization creates set-based contacts between areas identified by the connector projection. This is supported from both line and area connectors.

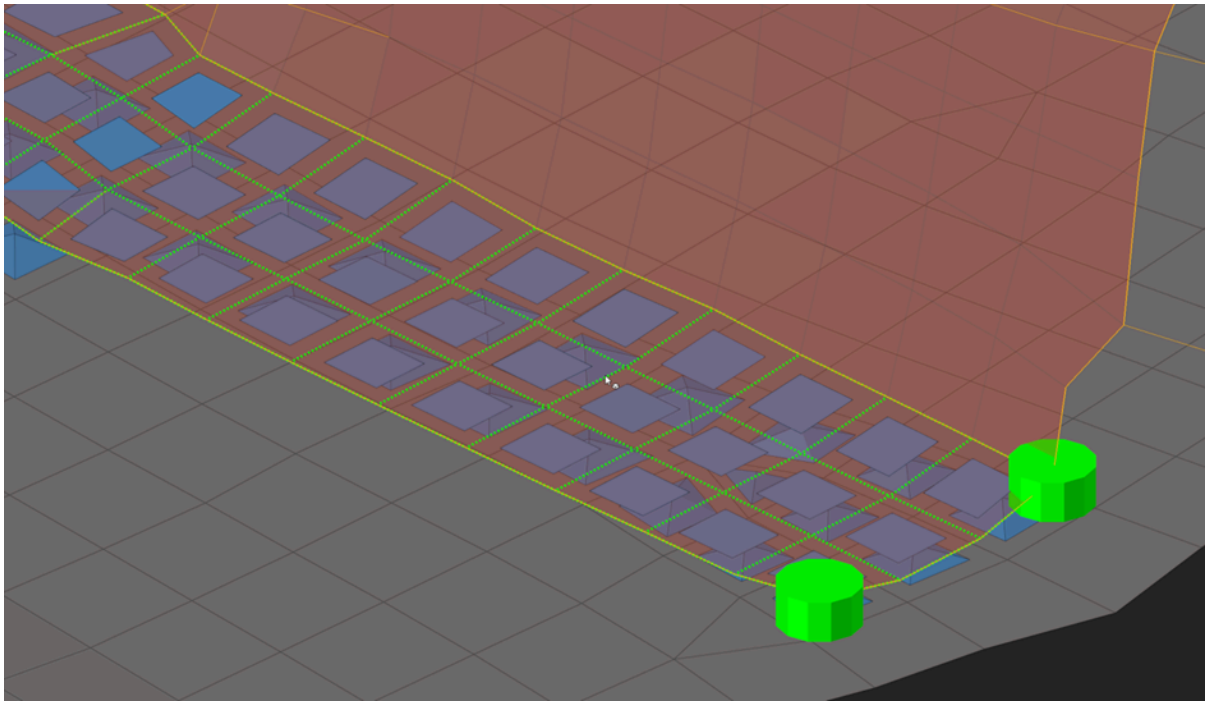

Figure 8:

Enhancements

Assembly Migration

Assembly links are now migrated to group links containing all of the components in the assembly.

Connector Graphics

The connector graphics for area connectors have been overhauled.
There should now be significantly improved frame rate when zooming, panning, and adjusting models with large numbers of connectors.

Miscellaneous

New attributes have been added to connectors to access more projection information for customization.
Improved direction recognition for Seam Quad when realizing on T edges.
Improved partitioning of the connectors over small gaps between parts.
Hemming has been renamed to match point connectors as RBE3(Load Transfer).
Add Links now remembers the previous settings from within a session.
An option added to organize the Connector to it's base part during creation.
Position connectors now works for all connector types.
Seam Quad realization now supports Tab/Slot type configuration.
ANSYS and PAM-CRASH realizations have been renamed to match the other solver profile realization naming convention.
RJOINTS and DOF have been exposed for fastener realization types.
An additional solid selector has been added to support point line creation.
Rebuild is now available as an option for area connectors.

Resolved Issues

- Hole type realizations are no longer failing with an open hole.
- Partially skinned solids are now treated as a single link during realization.
- The vector attribute on a connector now reacts appropriately to the Move tool.
- Solid fasteners now create a system when the option is selected.

Design Explorer

New Features

Configuration Design Variable
A new design variable type has been added which allows subsystem configurations to be swapped during explorations.

Enhancements

Generative Design
When filtering by concept, a summary table is shown, listing all runs in the selected concept family. The summary table provides options to load individual results for a selected concept.

Resolved Issues

- Unnecessary runs, due to lack of design variable combinations, have been removed from **Evaluation Status** dialog.
- When writing solver files only, a solver is no longer required to be specified.

Design Space

Enhancements

2D Voxel Face Elements
Create 2D voxel face elements will create a design space consisting of 2D shell elements only as opposed to 3D voxel elements. The respective PSHELL property and DTPL are created accordingly.

Extensions

Enhancements

Improved Version Administration
When multiple extensions with the same name and author but different versions and minimum product versions are registered, the extension with the most fitting `"minProductVersion"` and with the highest `"version"` will be loaded first.

```
<entry name="minProductVersion" value="2024.0" />
<entry name="version"           value="2.0" />

<entry name="minProductVersion" value="2025.0" />
<entry name="version"           value="3.0" />

<entry name="minProductVersion" value="2025.0" />
<entry name="version"           value="4.0" />
```

Figure 9:

Improved Messaging

Improved messaging when an extension fails to register. More information about other registered extensions with the same name and author are now provided.

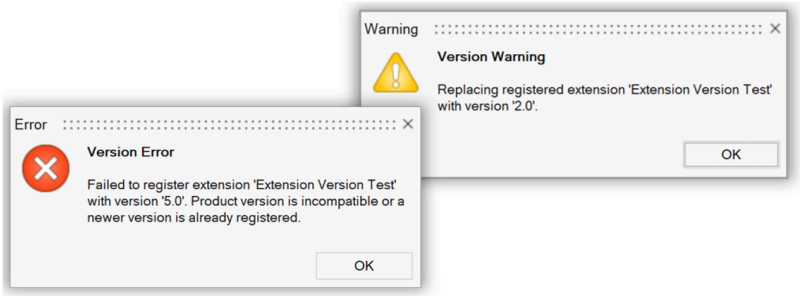

Figure 10:

$EXTENSION_DIR Variable Support

The `$EXTENSION_DIR` variable is now supported for all command attributes in the XML files. The previously limitation that the `$EXTENSION_DIR` variable was supported only for the command attribute no longer applies, all command type attributes are now supported.

```
<item tag="lineedit" type="combobox"
valuelistcommand="py: $EXTENSION_DIR.logic.comboValues()"
getcommand="py: $EXTENSION_DIR.logic.comboGet()"
setcommand="py: $EXTENSION_DIR.logic.comboSet()" />
```

Figure 11:

General

New Features

Diagnostics Utility

A diagnostics utility has been added to collect diagnostics files in the case of a crash and save them to a particular directory to be easily accessed.

Enhancements

Move Tool

Selection and manipulation are now separate actions with the addition of a new move button.

Picking a vector direction or plane normal (N1N2N3) to translate or rotate along is now easier with the support of the **Ctrl** click action within the Move tool.

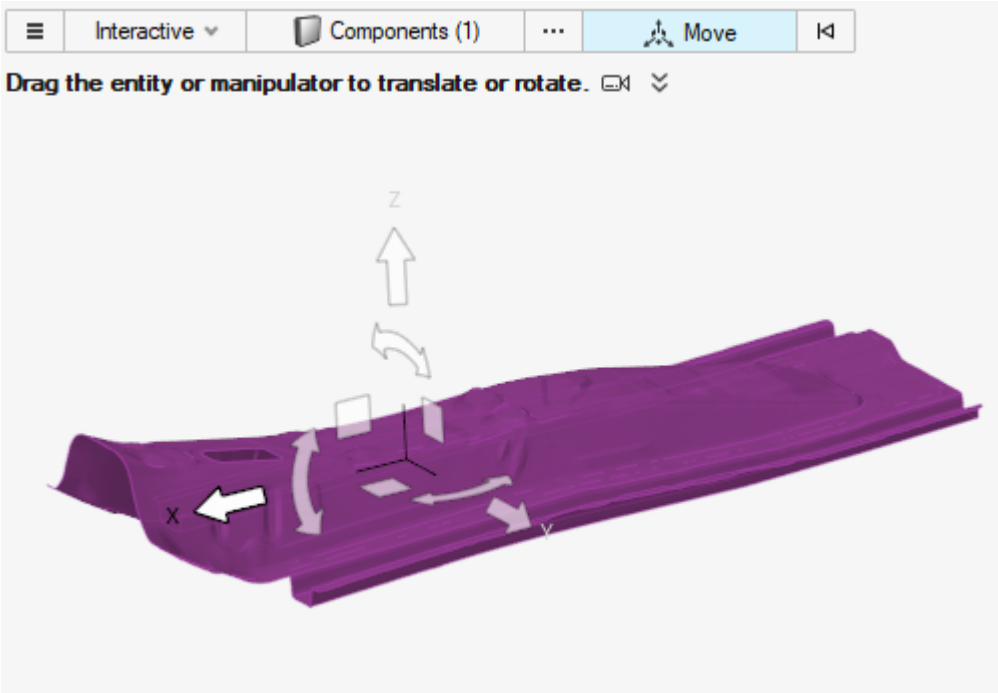

Figure 12:

Renumber

The text box has been updated to provide feedback where renumbering failed because the entities in the selection have their ID locked by the ID Manager.

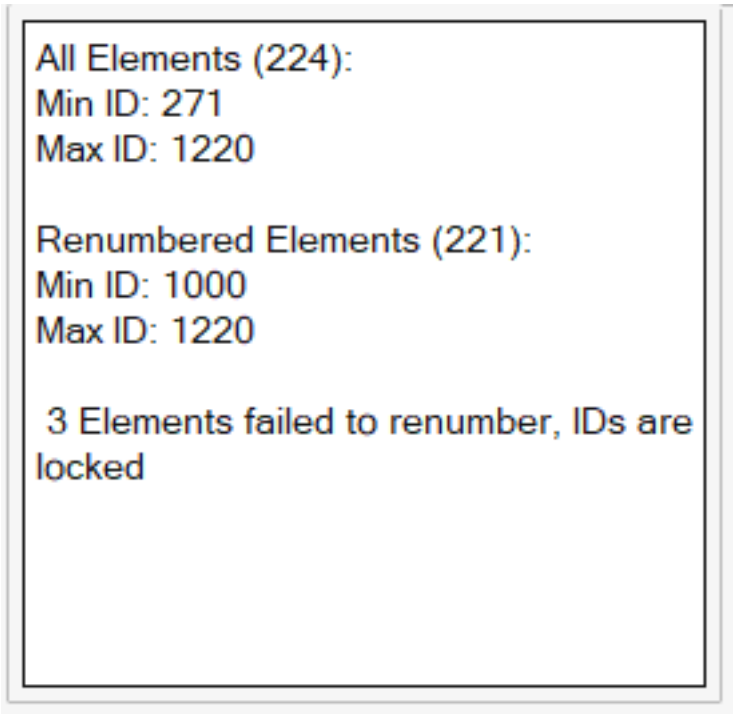

Figure 13:

The status message has been updated to provide feedback that, if some requested IDs were reserved using the ID Manager, that they were skipped as part of the renumbering.

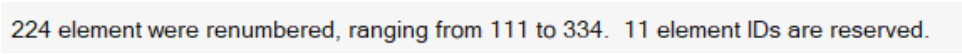

Figure 14:

An Organize option for nodes has been added in the right-click context menu.
The default Selection color is now automatically updated for a Dark Theme session.

Selection

The new By Type selection method can be used to select lines, points, surfaces, and solids by their topology type.
Part entity has been added to quick advanced selection (Alt).
The Sync with quick selection option is now remembered as a preference.
Keyboard shortcuts for segments and Vectors in the idle selector have been enhanced.

Sortable Tables in Online Help

The supported CAD readers have been reorganized into a sortable table. You can filter the table to show supported CAD readers by release or compare details for a single CAD reader across releases.

Resolved Issues

- The animation toolbar can now be resized when docked. Controls no longer visible due to resizing are accessible by clicking the chevron, which will appear to the right of the other controls:

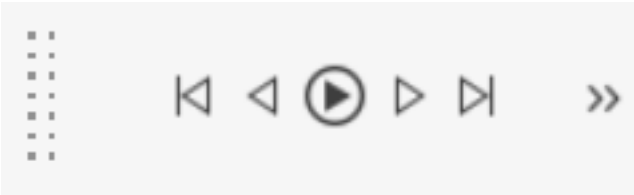

Figure 15:

- The incorrect cursor icon in the **File Import** dialog has been corrected.
- Reflecting Coordinate system using Transformation Tool now honors the base node.
- Ply shapes are now updated correctly.
- Deleting a non-active model from the Session Browser with a multi-session setup incorrectly no longer deletes the model from the active window.
- For view controls, doing a Show on a stored view no longer hides incorrect components.
- The **Ctrl** > **E** keyboard shortcut is now working.
- A different result is no longer given with nearby items depending on how many elements are selected.
- The search tool now correctly invokes the Move tool and Find tool options.
- Split by plane now manipulates the correct axis when using the Move tool.
- List of nodes no longer contains IDs from the wrong list when using by path selection.

Known Issues

- **File Open**/**Save** dialogs and Message boxes
  - There are inconsistencies seen in the appearance of the following types of dialogs:
    - **File** > **Open**
    - **File** > **Save**
    - Message boxes
  - Based on the available file extension filters in **File Open** and **File Save** dialogs, and how those dialogs or a message box are brought up, the appearance/style of the dialog can differ from other instances of those dialogs.
  - Some of these dialogs can go behind the main application when clicking elsewhere in the application, requiring you to bring them back to the foreground or reposition the application.
  - Example showing the different versions of the **File Open** dialog:

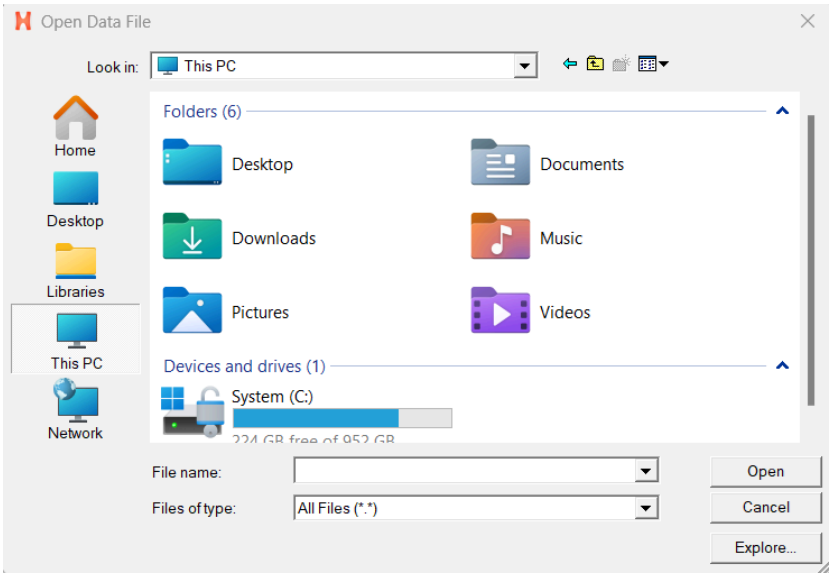

Figure 16:

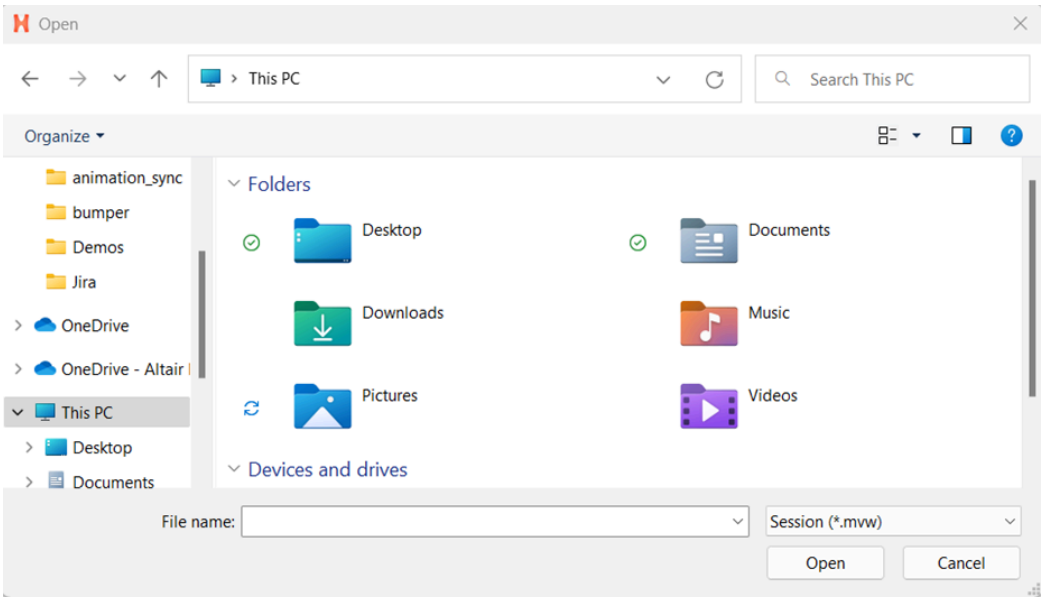

Figure 17:

- In multi-window layouts, longer guide bars may be truncated. To resolve the issue, resize the modeling window to be wider or expand the window so the guide bar is able to expand completely.
- In dark theme, thick white lines are seen between windows in multi-window layouts.
- The HyperMesh session can freeze if you open the **File** menu while the **Import Options** dialog is launching (but before it is displayed). If this happens, switch to another application and then return to HyperMesh to resolve the issue.
- On Linux, error messages can be seen in the terminal window after exiting HyperMesh, if the session has been left idle for some time.

- For Windows 11: On a multi-window page with two or more HyperMesh windows and at least one HyperView window, sometimes loading a model via the **File Open** dialog launched from HyperView's Load Model panel does not properly update the panel with the file name. This prevents you from loading the file.
  - If this issue is seen, you can load the file by copying and pasting, or directly typing the file path/name into the Open Model panel and clicking **Apply**.
- When HyperMesh is launched by double-clicking on a session (`.mvw`) file with multiple pages, in some cases not all the pages will be present in the HyperMesh session. To avoid this, launch HyperMesh via the Start menu or command line and load the session file from within HyperMesh via the GUI (menus, ribbons, and so on) or drag-and-drop.
- Occasionally, a new page created using the Session Browser context menu accessed by right-clicking on a page in the browser tree is not reflected in the page navigation tool. This can be avoided by creating new pages using the page navigation tool or by ensuring that the browser context menu used to create the new page is accessed via right-clicking on blank space within the browser tree area.
- On Linux, if a post client (HyperView, HyperGraph, MotionView, and so on) is shut down with the Session Browser closed, an application error is seen on relaunch and the Session Browser will not draw properly. Clicking through the error messages, closing the browser, and reopening it via the View pull-down menu will resolve the issue.
- In some cases involving multi-monitor setups, if a browser or dialog is dragged from one monitor to another, associated context menus can appear outside the browser/dialog, potentially even on the original monitor.
- When running in a NICE/DCV environment, there are drawing issues with the HWx Toolbelt and Advanced Capture.
- After switching from HyperMesh to one of the post clients (HyperView, HyperGraph, and so on), certain function keys not used as shortcuts in the clients can open the HyperMesh secondary ribbon with blank icons or exit the idle tool. To resolve this issue, switch to a tool available within the client and then exit the tool.
- HyperMesh can crash when publishing a session containing TextView and TableView windows to HTML.

Geometry

New Features

Imprint on Surface

Sketching now supports the Imprint on surface option for sketches drawn on a parent surface geometry.

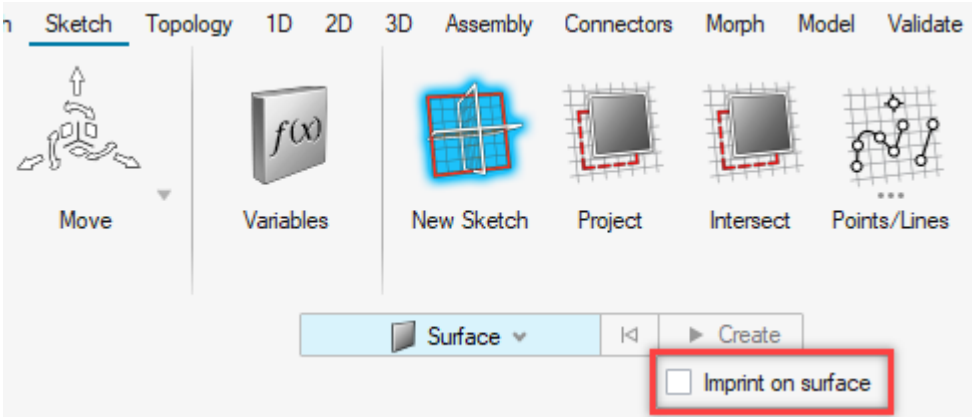

Figure 18:

FE Geometry for Solid Boolean
Solid Boolean now supports FE Geometry for Combine, Subtract, and Intersect operations.

Create Polyline From Node Cloud Data
Creating a polyline from node cloud data is now supported under the Line Creation tool.

Enhancements

Release Point Functionality
Release Point functionality has been added to the Quick Edit tools.

Align/Project Tool Improvements
The Align/Project tool has been improved for projecting on a plane for easy selection of the plane.

Meshing

New Features

Midmesh
A new method for midmesh has been added to handle extrusions. The two available options are:

- Casting: generate midmesh optimized for casting parts. The casting method handles midmesh on complex, freeform surfaces and variations in thickness.
- Extrusion: perform midmesh extraction on extrusion parts. The extrusion method handles midmesh on linear and consistent cross-sectional profiles.

Preferences
The Geometry and Meshing preferences are now combined under the Topology section.
A new preference has been added to choose between Mapped mesh or Freeform mesh when using surface creation tools.

1D Mesh
Section Manager is a new environment to create, edit, and visualize sections. It uses the same sketch graphics to visualize section topology and a dedicated browser for section data and computed properties. It is supported for sections of source type as Shape.

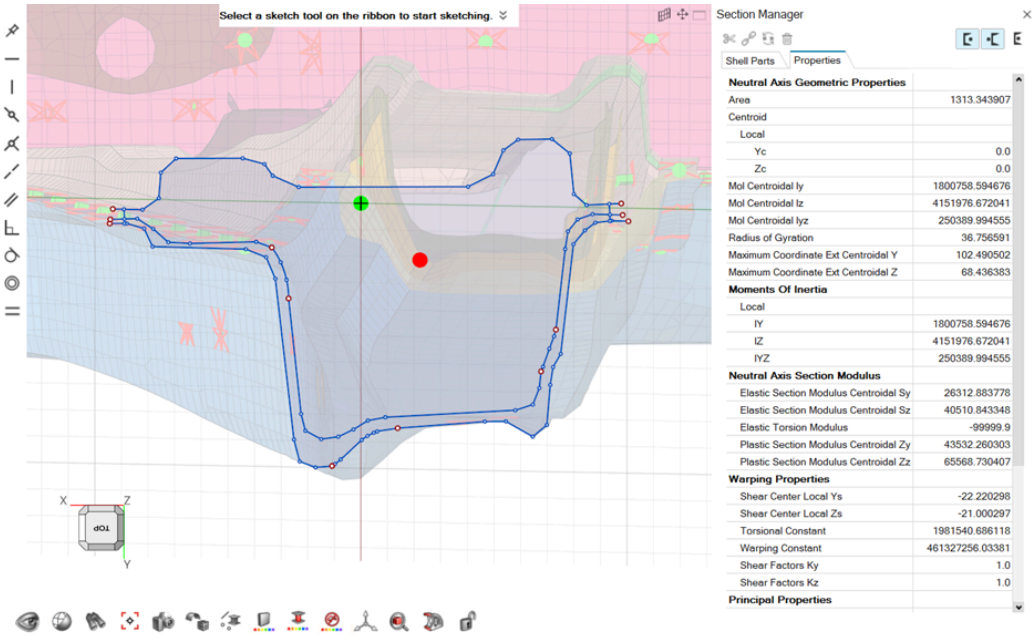

Figure 19:

Enhancements

Geometry Organization

The Create as new option in the Extend tool now follows the Create in setting under the Topology section for better organization.

The Create Points tool now adheres to the Create in setting under the Topology section for better organization.

Extrude

Selection of lines has been enabled in the target for hex creation.

Model Build

New Features

Team Part and Subsystem Library

Support for the new Team Part and Subsystem library has been added. The new library offers:

- Scalability to support a growing number of users, which enhances cross-team collaboration.
- No required IT involvement, which simplifies both setup and ongoing usage.
- An intuitive, easy to read folder structure for seamless navigation directly from the File Explorer.

Library Manager

The Library Manager dialog is enhanced to accommodate the Team library.

Enhancements

Metadata for Material
Metadata for material downloaded via AMDC is supported.

Classify Functionality
Classify functionality now is supported via API.

Custom Representation Control
Custom Representation Control now supports meshFlag.

Morphing

Enhancements

Miscellaneous
Advanced selection can now be accessed from the guide bar for Free and Proximity tools.
Support has been added for all window select types to work with Volume morphing.
The normal microdialog will no longer appear when trying to move 1D elements.
Selectors will now be cleared after an operation for all functions in all morphing controllers.
Domain creation has been improved for duplicate elements.

Resolved Issues

- The Remesh option now appears consistently when morphing using volumes.
- The QI legend now appears consistently when doing morphing operations with Element Check enabled after Remesh/Rebuild.
- Constraints are now created consistently when Mapping to Geometry with the Constraint Node option activated.
- The vector direction microdialog now appears after custom vector directions are defined.

PhysicsAI

Announcements

License checkout for prediction outside Altair GUI products, like HyperMesh or Inspire, will now draw 150 Altair Units.

New Features

New Architecture Introduced - Transformer Neural Simulator (TNS)
PhysicsAI now has a new AI training method - the Transformer Neural Simulator (TNS). This is in addition to the existing method, the Graph Context Neural Simulator (GCNS). Some key differences between the two:

- Usually, TNS should predict smoother contours than GCNS.
- TNS is less sensitive to variation in mesh sizes.
- On a GPU, TNS is faster than GCNS while on a CPU, it is typically the opposite.

Natively Reading Simulation Entities (Thickness and Material IDs)

PhysicsAI can now extract thicknesses and material IDs if there are solver decks available along with the training files. These decks should be in the same location and have the same name as the corresponding solver file. For example, if a file called `Ibeam.h3d` has a file accompanying `Ibeam.fem`, then the thicknesses and material IDs can be parsed by selecting **Extract Simulation Properties**.

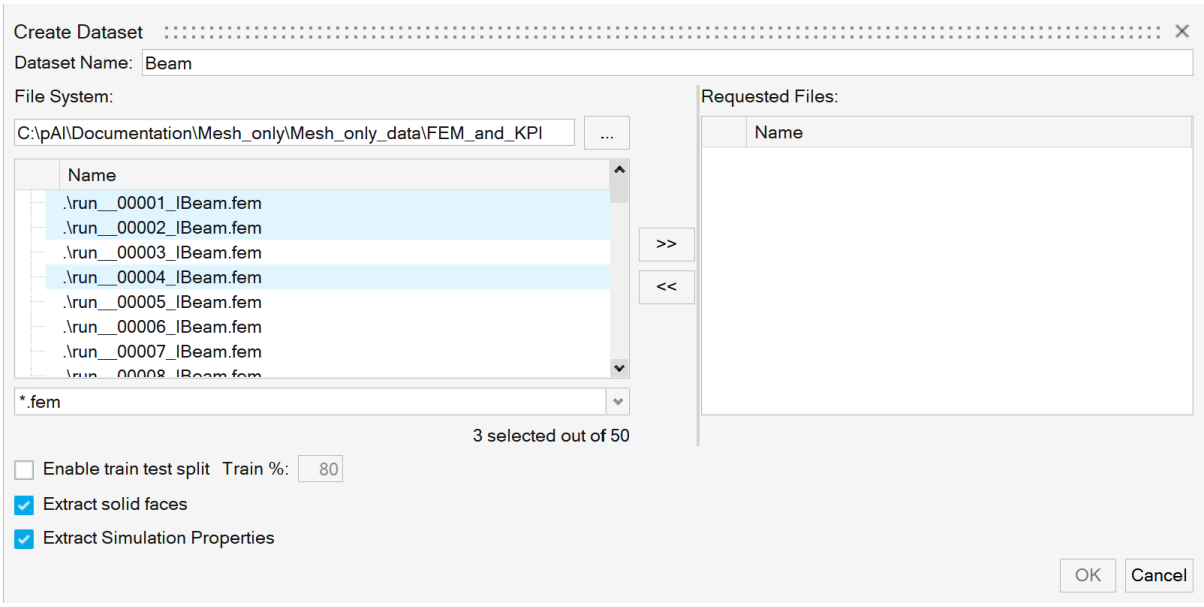

Figure 20:

The list of supported solver decks include: Optistruct, Radioss, LS-DYNA, Nastran, Abaqus, and ANSYS.

Enhancements

Mesh Alignment for Translational Invariance

PhysicsAI is sensitive to meshes being translated and rotated in space. Earlier, you had to manually orient the meshes to eliminate this source of noise. Now, using the Mesh Alignment feature during model training, the meshes can be adjusted during training, testing, and prediction. The meshes are aligned such that the center of gravity is coincident at a common point. This option can only correct translational variances and not rotational variances.

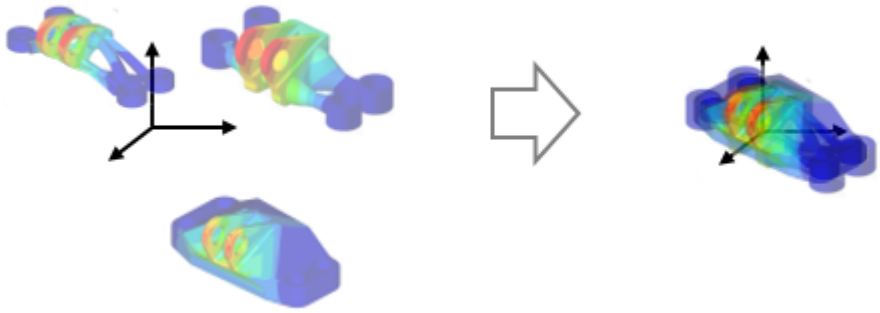

Figure 21:

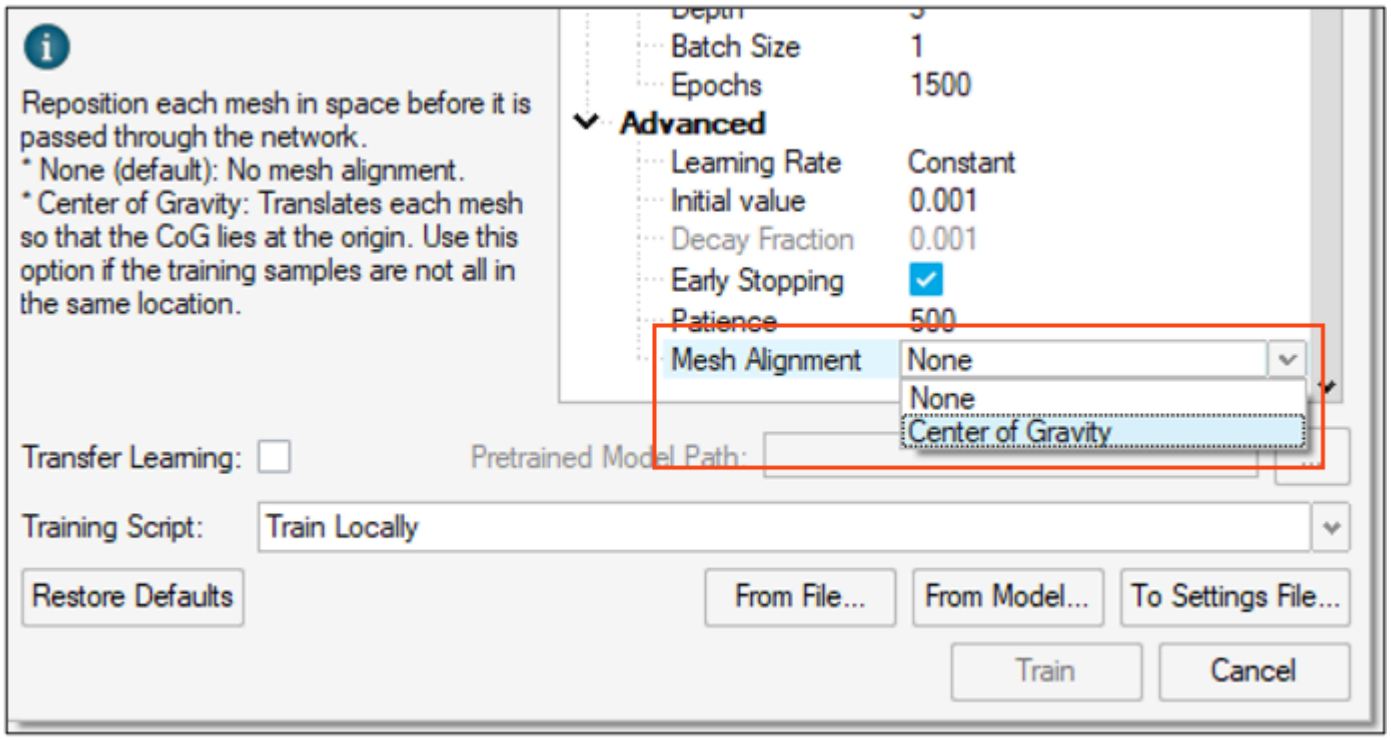

Figure 22:

Enhanced Dataset Visualization and Outlier Detection

Previously, you had to utilize other tools, such as HyperStudy, to curate the data and identify outliers. Now, outliers in the dataset are identified based on the you select. A Z-distribution is fitted to the data and points that fall in the 3-sigma tails are highlighted as outliers.

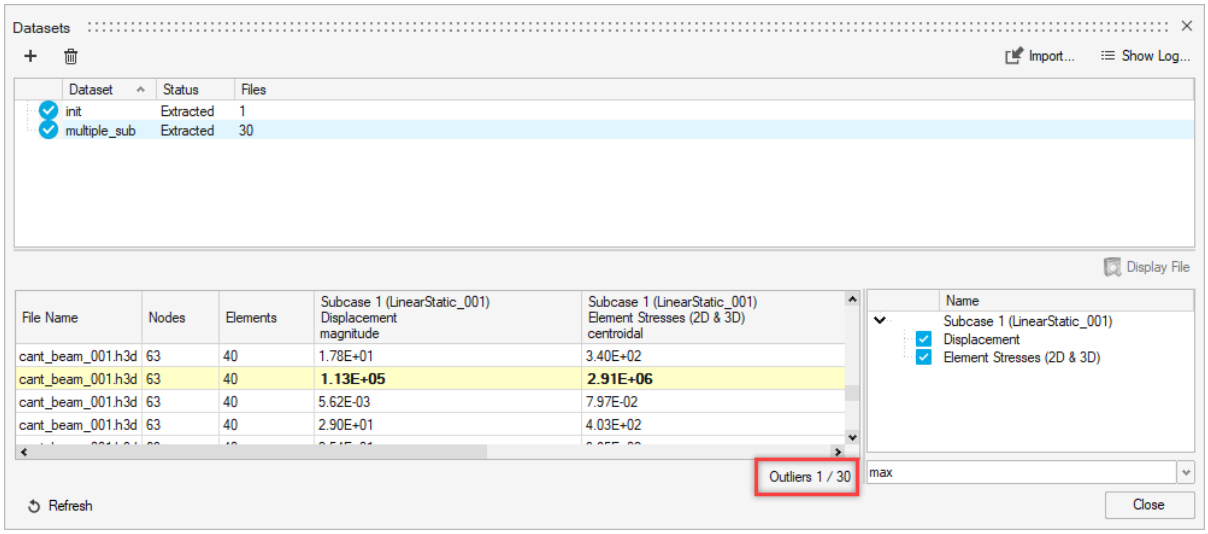

Figure 23:

Similarity Score in the PhysicsAI Connection in HyperStudy

You can now access the Similarity score for a prediction made using a physicsAI model in HyperStudy. This can be a useful response to qualify results based on expected accuracy. For example, using it as a constraint to reject predictions with low Similarity scores. The Similarity score is automatically added as a response if it is available in the selected physicsAI model.

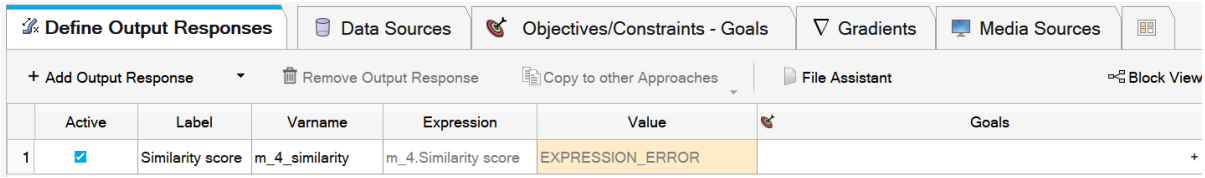

Figure 24:

Known Issues

- Predictions made in the Radioss solver profile within HyperMesh may not function correctly when model features are used (like thickness or material). This issue can be avoided by using another solver profile, such as OptiStruct.

Resolved Issues

- Eroded elements are now hidden during visualization. Previously, eroded elements resulted in an exploded mesh resulting in confusing visualization. Now, elements are hidden after the point of erosion.

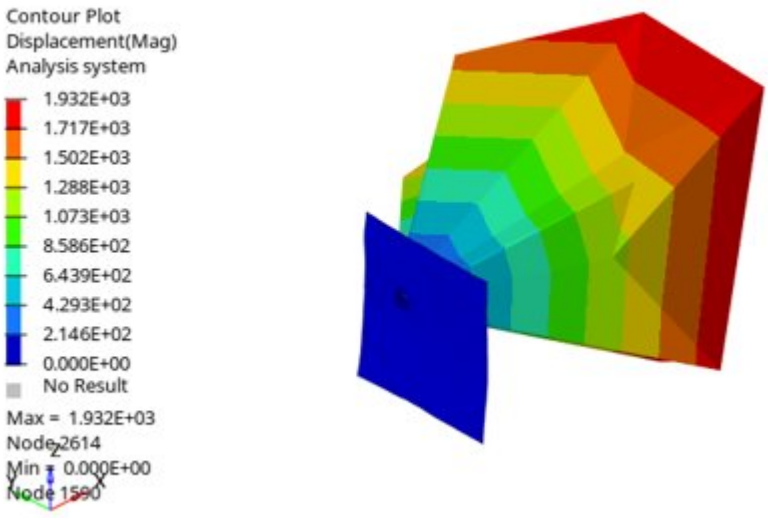
Contour Plot
Displacement(Mag)
Analysis system
1.932E+03
1.717E+03
1.502E+03
1.288E+03
1.073E+03
8.586E+02
6.439E+02
4.293E+02
2.146E+02
0.000E+00
No Result
Max = 1.932E+03
Node 2614
Min = 0.000E+00
Node 1590
Z
Y
X

Figure 25:

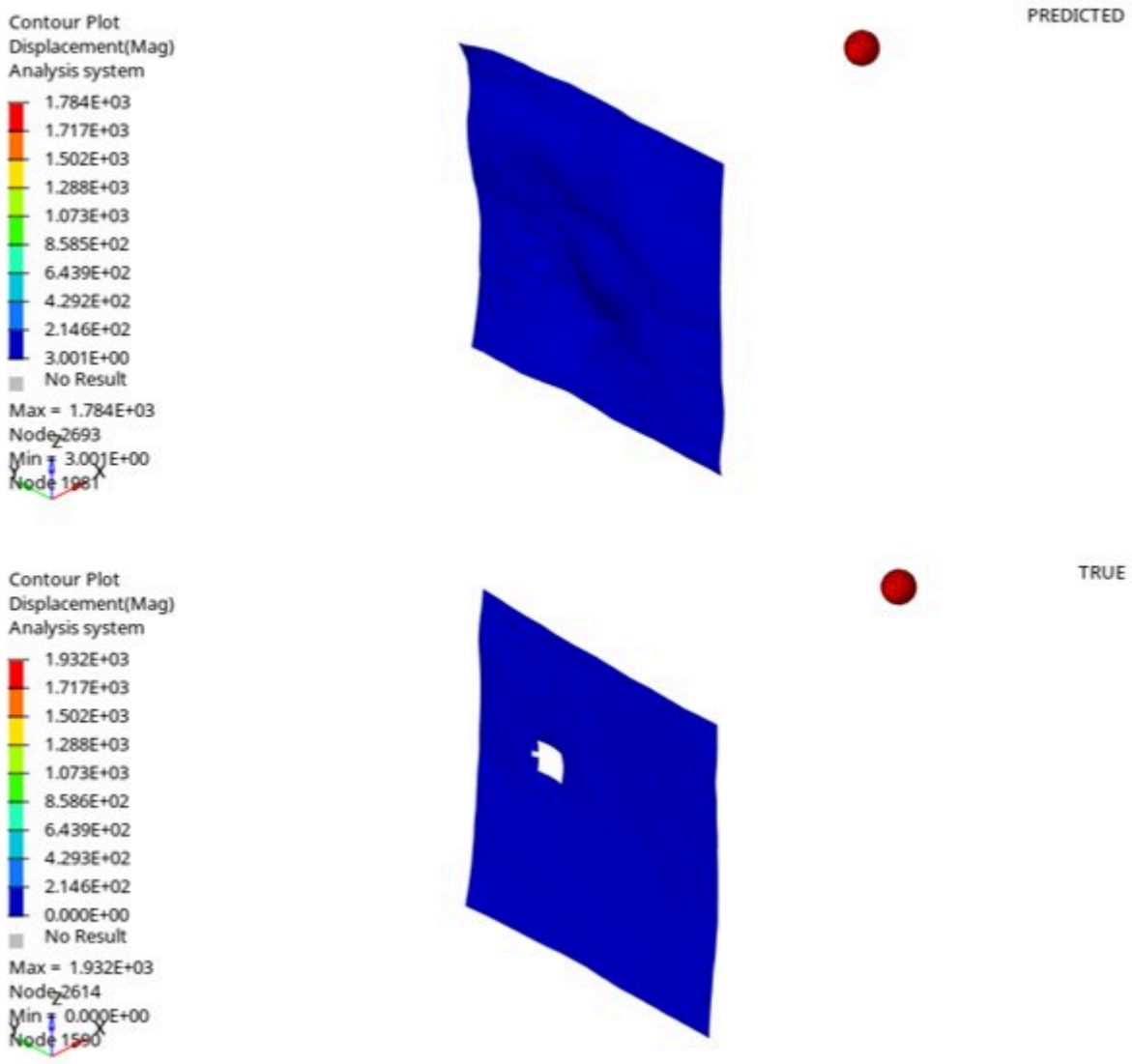

Figure 26:

- Previously, errors were encountered if the training data contained parts with no results. Now, physicsAI automatically excludes such parts and the training can continue.

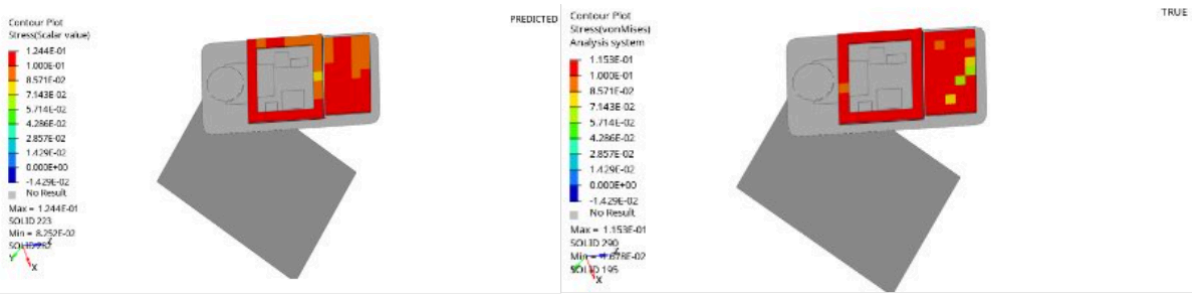

Figure 27:

Plots

No updates were made to Plots in 2025.

Post

Enhancements

- Support of vector and tensor plot during animation.
- Vector and Tensor plots can now be shown on undeformed or on deformed shapes.

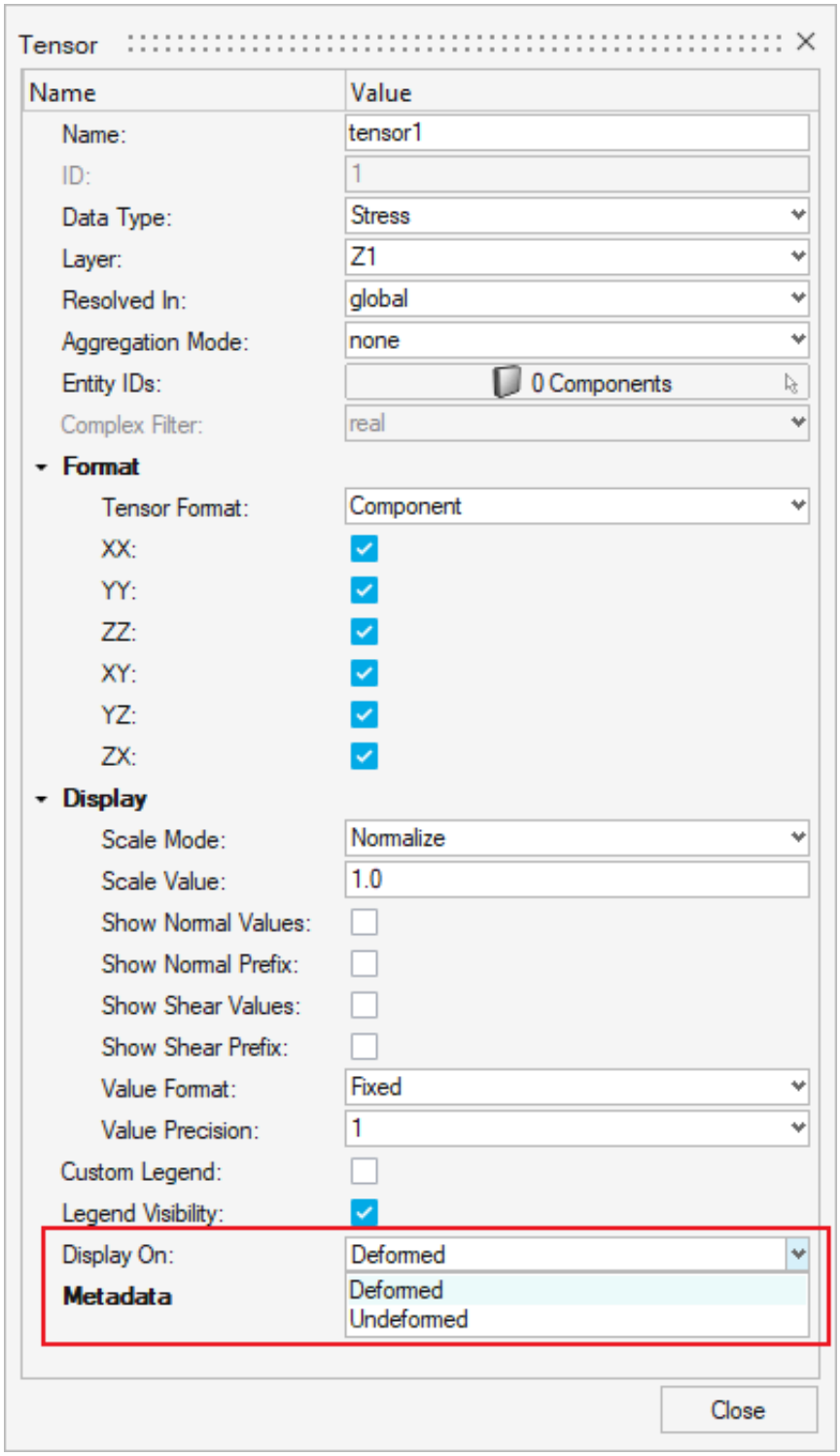

Figure 28:

- A small/large deformation switch has been added to subcase entity.

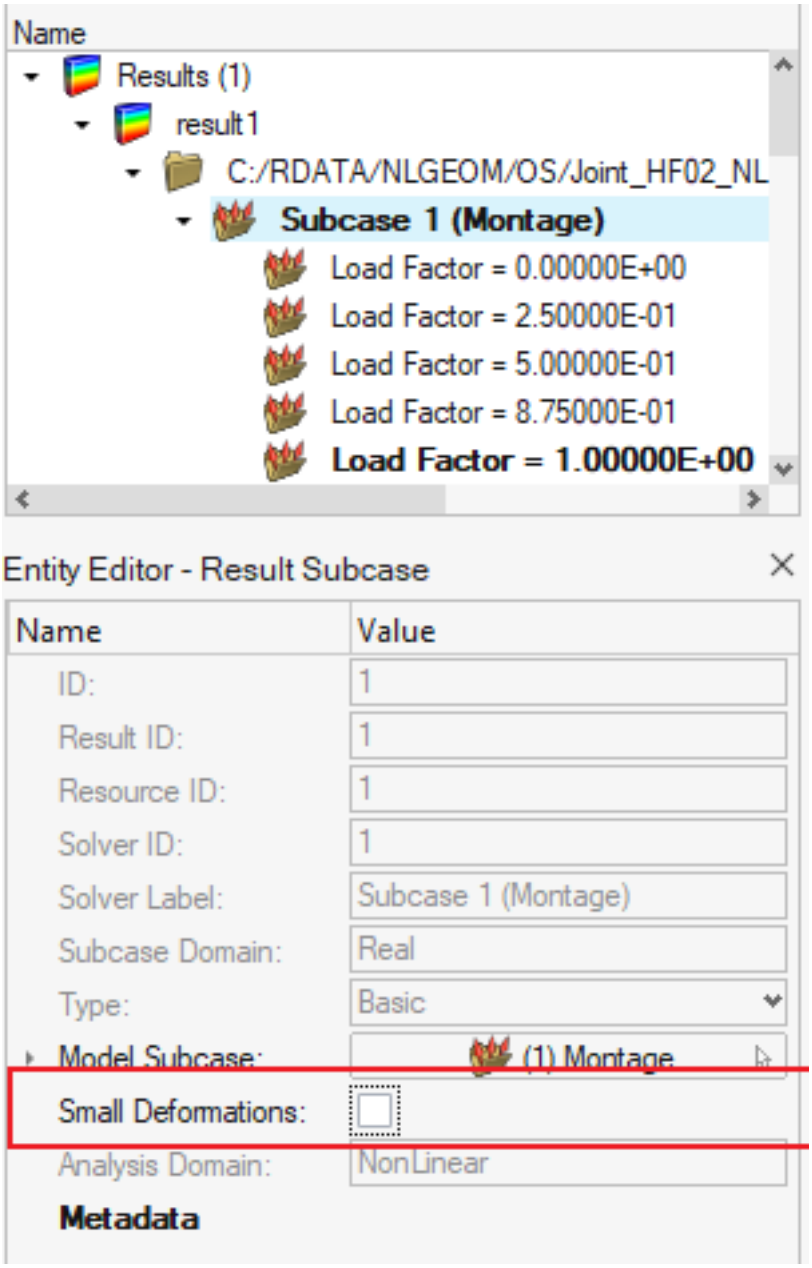

Figure 29:

- The Small Deformation check option has been added in the post preference page to control results query accordingly.
- A new tool has been added to show post preferences. From the Post ribbon, select the **Post Preferences** tool from the Results tool group.

Figure 30:

- Closed form complex filter calculation is now available for Von Mises Strain invariant.
- Rainbow scheme is now set as default for all plot control legends.
- Modal Percentage is now exposed as deformation mode.
- A change has been made in labeling for averaging method to Aggregation Mode.
  - The former set up as average = simple is now relabeled as average then compute.
  - Mode compute then average has been added.
- Aggregation based on part boundaries has been added as an option to plot control.

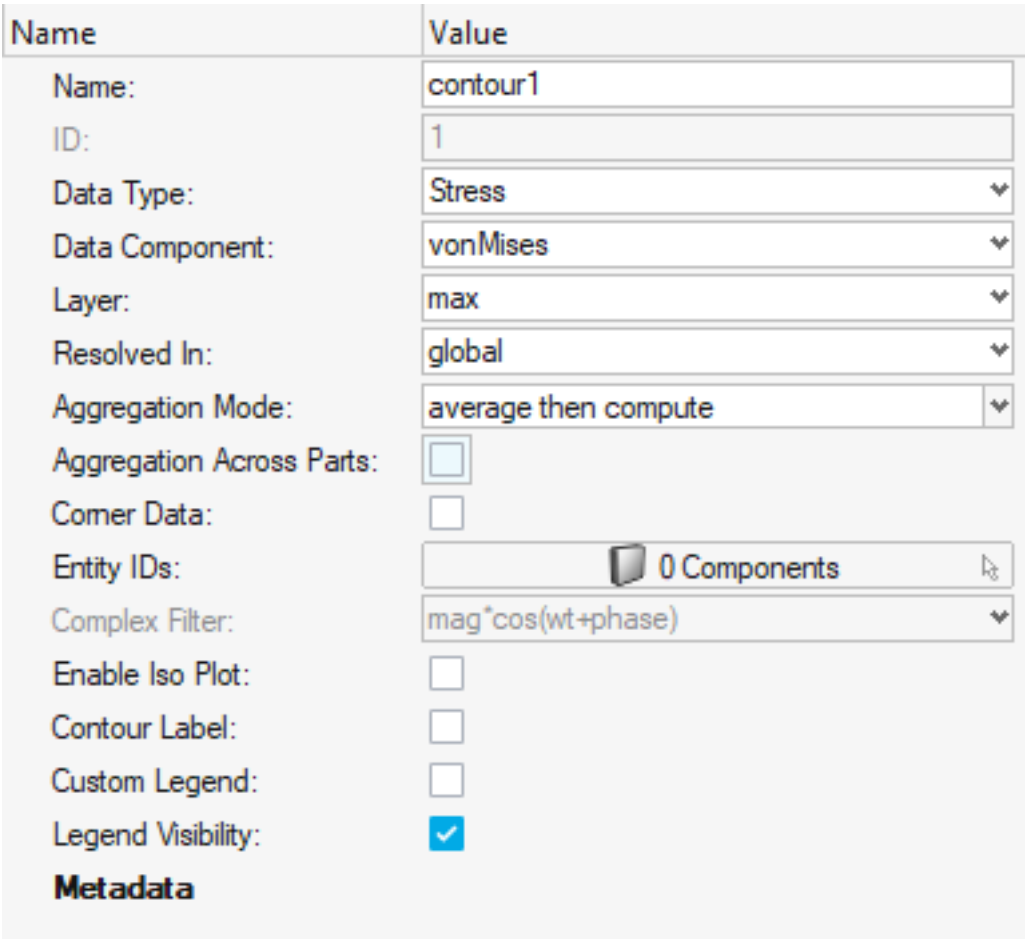

Figure 31:

- An `*.avi` and `*.gif` export option has been added from animation in HyperMesh.

Known Issues

An application crash is possible when importing another model/solver deck after having imported results. This was already an issue in 2024.

Resolved Issues

- An issue has been resolved with w.r.t. csv configuration file for generation of derived load cases when both LSP and Envelope cases were defined.
- The *tablecontour API now triggers full clean up of any displayed plot control from result workflow in HyperMesh. This will avoid any display issue where plot control view mode was not cleaned and any plot based on table was not displayed.
- The marker plot now shows all available layers.

Safety Report Manager

Highlights

- Altair Safety Report Manager is capable of generating reports for Seat Impact load cases.
- Radioss and PAM-CRASH solver support is added for all structural modules across frontal, rear, and side impact regulations.
- The following standard modules are now included:
  - Altair Time History Plotter (ATP)
  - Contour Plot
  - Animation New
  - Measure Plot

New Features

Seat Impact Load Cases

The following seat impact load cases are now supported:

- Whiplash load case for EURO-NCAP, CNCAP, JNCAP, and KNCAP regulations.
- Energy Dissipation and Static Headrest test load cases for AIS 016, ART 18b, ECE R17, FMVSS 202a -5, GB 14167, and KMVSS 103 regulations.
- The Seat Belt Anchorage Description module is now available as part of the ECE R14 regulation.
- The Luggage Retention module is now available as part of the ECE R17 regulation.

The following new standard modules are included:

- Seat bolt recliner moment
- Floor bolt force
- Belt forces

Occupant Modules

The following occupant modules are now supported:

- Frontal impact - UN Regulation137 Rigid Barrier In-Position
- Frontal impact - UN Regulation94 Deformable Barrier In-Position
- Frontal impact - EuroNCAP - MPDB

Enhancements

Frontal Impact Structural Module

The following enhancements have been made to the Frontal Impact Structural module:

IIHS - SORB Intrusion Module

This module is enhanced to support 3D animation results data. The SORB Intrusion module now supports both time history data as well as 3D animation results data.

IIHS - ODB Intrusion Module

This module is enhanced to support 3D animation results data. The ODB Intrusion module now supports both time history data as well as 3D animation results data.

MPDB Intrusion Module Improvements

The following improvements were made:

- Added the capability to create an intrusion grid pattern.
- Allow editing of the regulation-specific limits for Standard Deviation (SD) and OLC values in the modifier plot.
- Included the evaluation area for the bottoming-out picture.
- Added the barrier intrusion value in the spreadsheet.

Generate Run Script

This feature is now included. This generates a Windows batch file (`.bat`) and Linux shell script file (`.sh`), which can be used to run Altair Safety Report Manager in complete batch mode (non-GUI) for both the single and overlay run modes.

Added the Best View Capability to the Plastic Strain and Displacement Contour Modules

Added the capability to control the sequencing of modules in the final PowerPoint (PPT) and HTML report.

Known Issues

The following known issues will be addressed in a future release as we continuously improve performance of the software:

- Input validation check is not supported for the Battery Section Force and BOM modules.
- The Load Path, Run Statistics, and Weld Failure modules are not supported for the Radioss and PAM-CRASH solvers.

Resolved Issues

The issues related to the following modules have been addressed.

- Application error with Engine Mount Failure.
- In Run statistics, the solver version is missing due to format change in solver message file.
- Entity type label missing in the BOM configuration file.
- Error in the Vehicle Kinematics vertical and the Structure Plastic Strain.

Safety Tools

Enhancements

Pedestrian Pre tool

Update of EuroNCAP ver 9.1 with Test to roof marking.
Update of GTR/UN-R127 with Test to windshield.
Addition of JNCAP.
Ability to include ground plane inclination while positioning impactors.
Impactor position based on Local coordinate system.
Enhanced marking logic for Windshield Rear Reference Line for roof.

Skeleton Modeling

New Features

Categorize Tool

The Categorize tool has been added. The Categorize tool is designed to streamline the development of reduced order models by automating the detection and categorization of subsections with an automotive model.

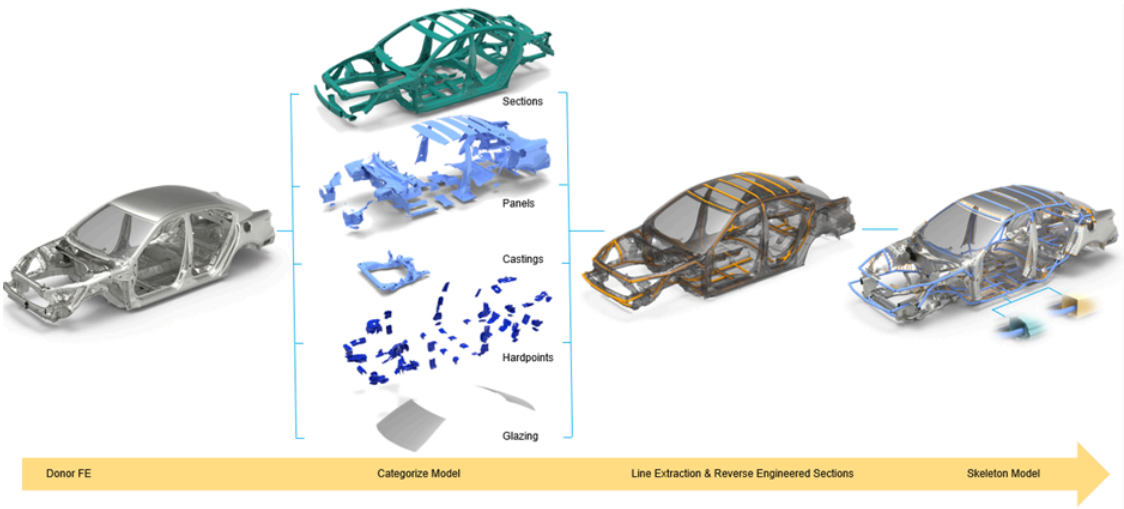

Figure 32:

Extract Line Tool

The Extract Line tool has been added. The Extract Line tool generates a mid-line representation from the selected input, which is essential for creating reduced-order models.

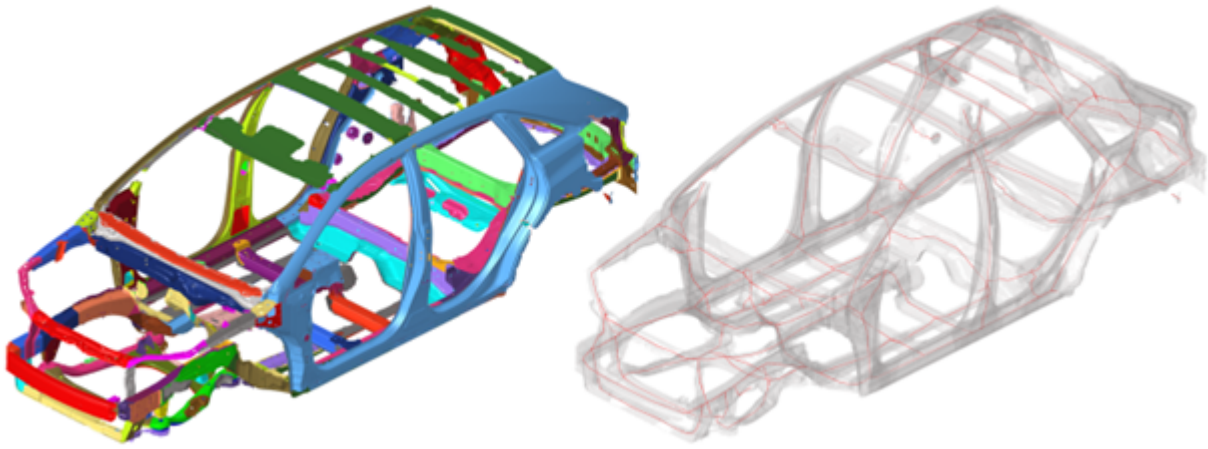

Figure 33:

Auto-Offset

Auto-Offset restores the position of members after associated sections are created.

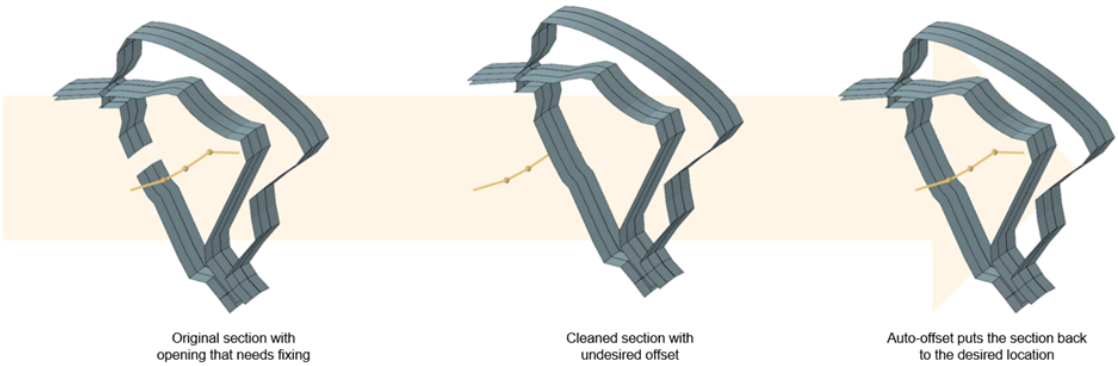

Figure 34:

Enhancements

Section Cleanup

Section cleanup will remove overspill areas of a section to reduce the section to only what's desired. The option will remove a large amount of manual cleanup.

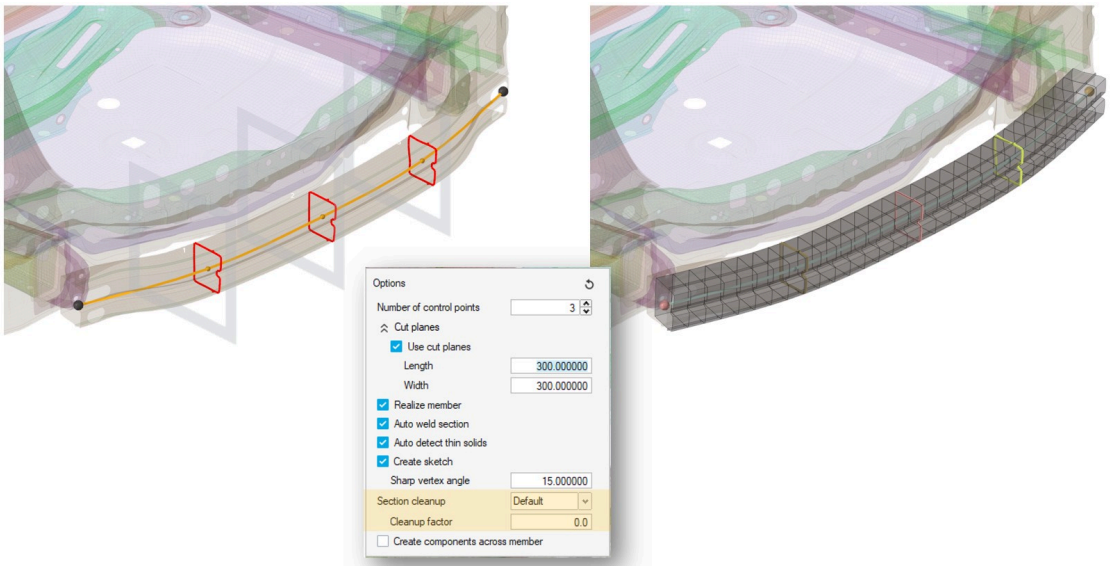

Figure 35:

Solver Conversion

Resolved Issues

Abaqus to OptiStruct conversion improvements have been made on models with large number of constraints.

Task Manager

Enhancements

Process Editor

In the Process Editor, the Save button has been modified and provides two functionalities: Save and Save as.

Save	Saves the newly created process to an XML file or updates the current XML file with the new content.
Save as	Saves the process to an XML file different than the current one.

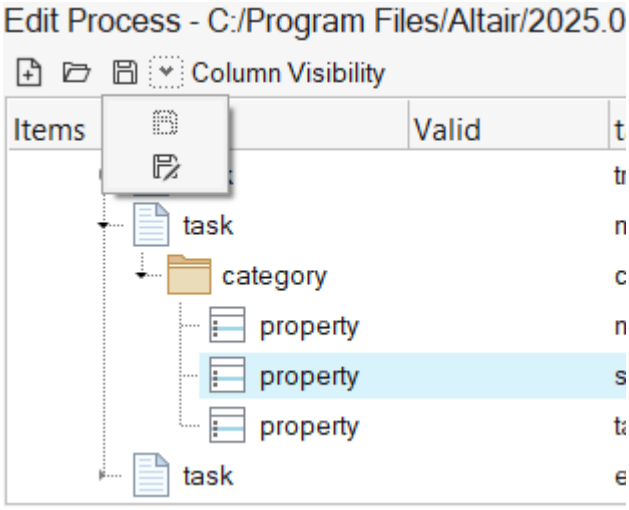

Figure 36:

Resolved Issues

The *hmmultiselector* widget property was not respecting the definition in the XML file, and the component entity type was always included in the list of the selectable entity types. The widget now correctly offers only the entity types specified in the XML file.

CAD and Solver Interfaces

Abaqus Interface

Enhancements

LARC05 Damage Model
A new LARC05 damage model has been added for composites.

STRUCTURAL Option Support
For the *DAMPING keyword, the STRUCTURAL option is now supported for Abaqus/Explicit profile.

Resolved Issues

- The SPH tool is now available in search for Abaqus/Explicit profile.
- Nodes with system assignment are no longer lost on import.
- An issue with reading *SHELL SECTION and *SHELL SECTION, COMPOSITE has been resolved.
- A warning message has been added if an Abaqus parts and instances deck is exported in flat format.
- A solver keyword for *COUPLING and *KINEMATIC/*DISTRIBUTING has been added to the Entity Editor.

ANSYS Interface

New Features

The TB(JOIN) card is now supported for stiffness, friction, and damping.

CAD Interface

New Features

- NX 2406 support has been added (Native reader).
- 2024 3DXML support has been added.
- Step AP242 XML is now supported for BOM Export.

Enhancements

- Inspire Parameter update is now supported on Linux.
- The CAD Reader log file extension has been changed from `*.msg` to `*.log`.

OptiStruct Interface

New Features

Encryption Functionality

Encryption functionality is now available in OptiStruct input files to encrypt confidential information.

- Properties and materials can be imported and exported with encryption.
- Damage and table cards associated with property and material entities can also be encrypted.
- Additionally, if the supported entities are defined within an INCLUDE file, the entire INCLUDE file can be encrypted.

Enhancements

Mass Calculation of NSM

Mass calculation of NSM defined through SETs with property.

Known Issues

- SUBCASE cannot be selected for ERP constraint.
- NU option cannot be cleared once populated on DREPORT card.

PAM-CRASH Interface

Enhancements

PAM-CRASH 2024 Profile Support

Support of the PAM-CRASH 2024 profile has been added.

Radioss Interface

New Features

- A new Initial State entity has been introduced and supports all the following keywords: /INISHE, /INISH3N/, /INIBRI, and so on.

Enhancements

- A new Model Checker check has been added for RBODY in component with Part card image instances with auto correction.
- Mass calculation has been improved in the Validate and Mass Measurement workflows when a lumped mass is applied on a free RBODY secondary node.

Known Issues

- After switching from the OptiStruct to Radioss solver profile, the RBODY node set is now correctly created on export.
- Auto correction of the Model Checker check for RBODY main node on element instances now works for two nodes RBODY type.

Altair HyperMesh CFD 2025 Release Notes

Highlights

HyperMesh CFD 2025 introduces the following new features:

Geometric Modeling:

- Support of all critical discrete, Geometry ribbons tools in template manager

Case Setup (External Aerodynamics):

- Ground patch visualization

Post-Processing:

- 3D hydrodynamic acoustic map computation in the GSP tool
- Export option for time-dependent data written to VTKHDF
- ISO standard center frequency for octave and fractional octave output
- Additional supported files from the **Import/Open** menu

General resolved issues:

- Crash on loading HyperMesh CFD if admin installed
- Template manager dialog closed when pressing **esc** on operation definition dialog
- View toolbar was not visible on opening session
- **File** > **Save** not working if changes were made in case setup

Geometric Modeling

New Features

Template manager updates
: Supported all tools from discrete ribbons, critical tools from geometry ribbon, scale, mirror, move operations in template manager. Categorized operations for ease of use.

Enhancements

Drag/Extrude tool for lite modeling
: Now supports Drag/Extrude tool for face creation by dragging nodes in desired direction. This will help to create faces before wrapping.

Right click options consolidation
: Consolidated right click options for all geometry repair ribbons and make it work for Mixed model type.

Resolved Issues

- The template manager dialog closed on pressing **esc** on operation definition dialog

- Template manager Delete operation deleted only one row
- Template manager wildcard (*) entry was not working
- Parts grouping was broken after running wrapping or leak check
- Parts are broken after fill hole in lite model

Case Setup

New Features

Ground patch visualization

The ground patch around the vehicle is now graphically displayed on the wind tunnel floor when the "**Enable ground patch**" option is enabled.

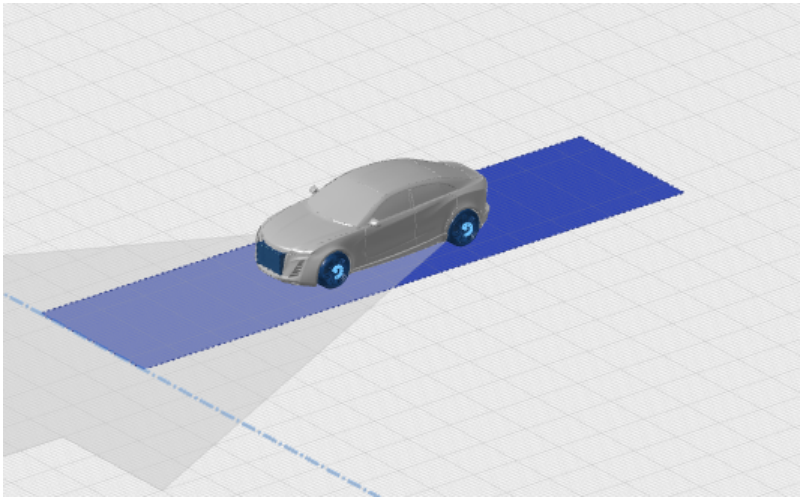

Figure 37:

Resolved Issues

- XML issues:
  - Turbulence offset was not editable
  - Wheels were misidentified as body parts
  - Merge output files option was written incorrectly
- Attribute sync issues:
  - Mesh refinement for OSM/MRF volumes should be in sync with property editor
  - Scaling factor gets reset on closing run dialog
  - Fan can be accidentally setup twice leading to mismatch in properties when reset to body
  - Number of layers in offset refinement is not refreshed
  - Visibility of turbulence source without mesh control is reset
  - Virtual fan coefficients get reset without enter key
- Multiple database imports no longer create multiple wind tunnels
- Warning added when deleting run from run dialog

- Fixed segmentation errors:
  - On fan axis reset using axis buttons
  - Fan axis reset on tab/enter keys
  - On selecting “Identify” part as body

Post-Processing

New Features

Provide capability to compute 3D hydrodynamic acoustic map in GSP tool
Provides the capability to compute three-dimensional hydrodynamic acoustic maps from the GSP tool. The existing GSP functionality is primarily focused on surface data and computation of SPL (dB), this improvement seeks to extend it to handle volumetric pressure or velocity data to compute the power spectrum.

Add "Export" option for time-dependent data
Provides the addition of an "Export" option for time-dependent data in HMCFD written to VTKHDF. This feature enables users to create an extract database from transient datasets, which can be loaded separately for faster interaction and manipulation.

ISO standard center frequency for octave and fractional octave output
Modifies the center/min and max frequencies for GSP when band type is octave or fractional octave. ISO standards have been applied to HyperMesh aeroacoustics results output.

Add supported files from Import / Open
Provides the addition of VTKHDF (.series) and Ensight Gold result files to the list of supported file types by group for results import.

Resolved Issues

- Export to h3d when time-dependent data is loaded now retains correct timestep data and range
- HMCFD Report: Streamlines, *seed_type=rectangular* has been updated to include each cartesian direction
- Session no longer crashes when loading specific .h3d file created by Physics AI
- AcuSolve Log Reader options with “Derived Quantity Output” no longer gives active exception when results are not available

Solution

New Features

No changes have been introduced to CFD Solution in HyperMesh CFD 2025.

Known Issues

For more information on issues found throughout the release, and their workarounds, please see the Altair Community.

Altair HyperView 2025 Release Notes

General

Highlights

- Support for additional solvers, animation modes, and multiple result files in the HyperView - MultiCore profile.
- GLB Export - Support for legend variations and skipping model hierarchy.
- Python API's for querying results, interactive selection context, 3D export and sets.
- Safety Tools - New Upper Interior Impact Post Tool

New Features

- **Support for additional solvers and animation modes in the HyperView - MultiCore profile**

  Coverage in the HyperView - MultiCore (HV - MC) profile has been greatly expanded in this release with addition of linear animation mode and support for OptiStruct, Nastran and Ansys results. Previously, HV - MC supported only transient animation mode and Radioss, LS-DYNA, Abaqus, and PamCrash results. Additional enhancements to HV-MC are planned for upcoming releases.

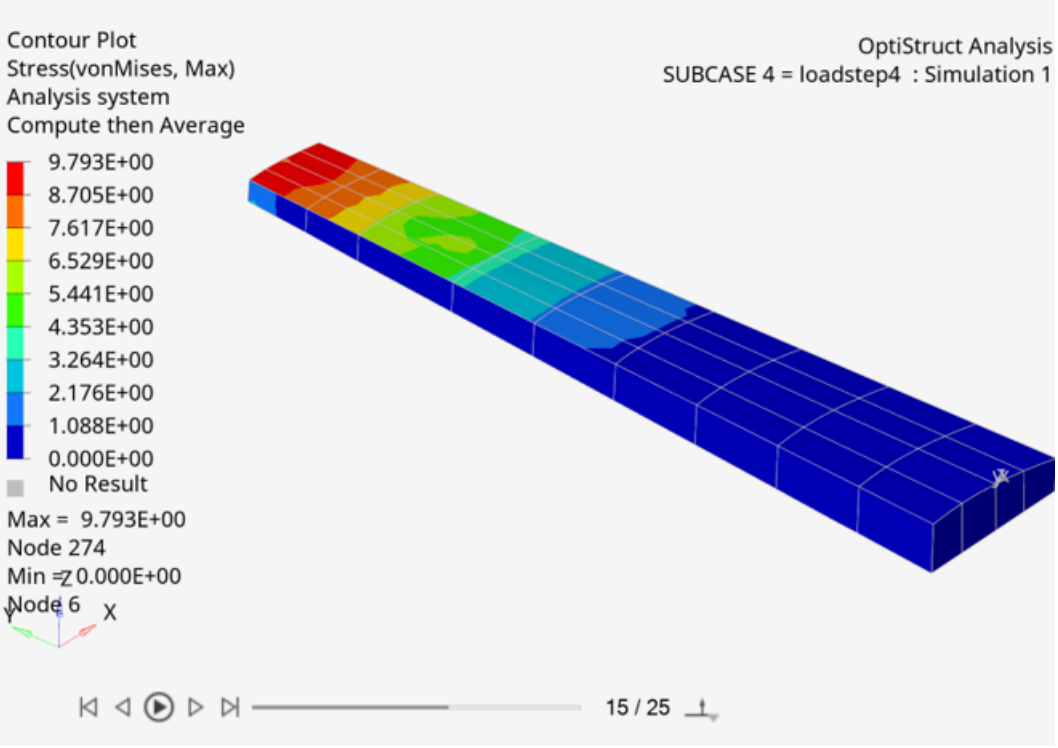

Figure 38:

Also, HyperView - MultiCore now supports the option to attach several results files to a single model. Previously this was available in the HyperView - General profile only.

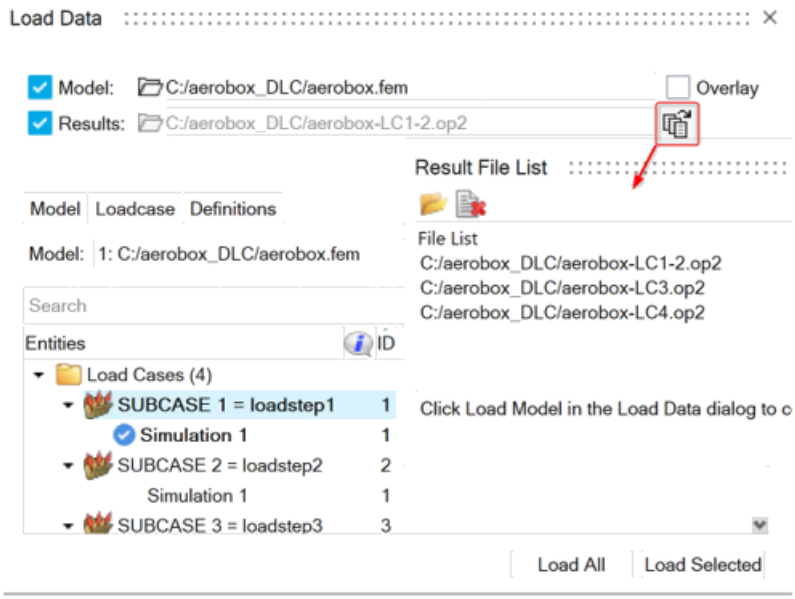

Figure 39:

- **GLB Export Enhancements**
  - You can now skip model hierarchy during GLB export resulting in faster export times and smaller file sizes. This option is relevant for both 3D Model and PPT formats of viewing GLB apart from viewing modes that do not use model hierarchy. The effect of this option will be more visible for models with a high number of parts. The model hierarchy will be skipped from export as default.

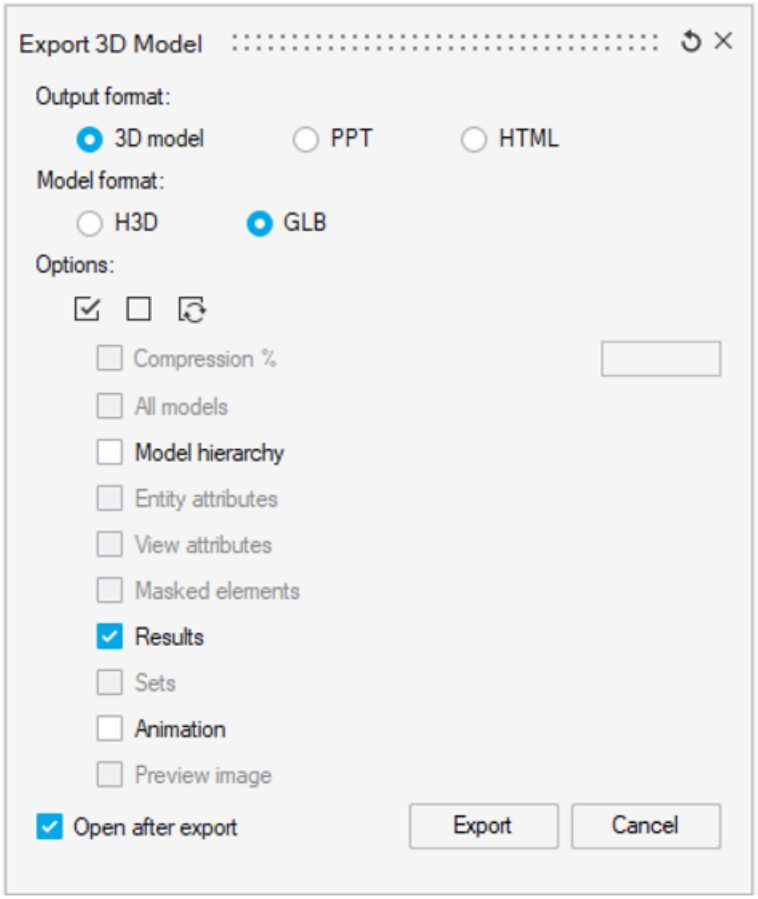

Figure 40:

- GLB supports legend variations in HyperView contour plots such as custom Max, Min and intermediate values, as well as the log interpolation method.

- **Python API**
  - The new QueryResultsTool() class has been introduced. This API allows you to query entity attributes in a NumPy array format, all entity attributes from the query table tool are supported.

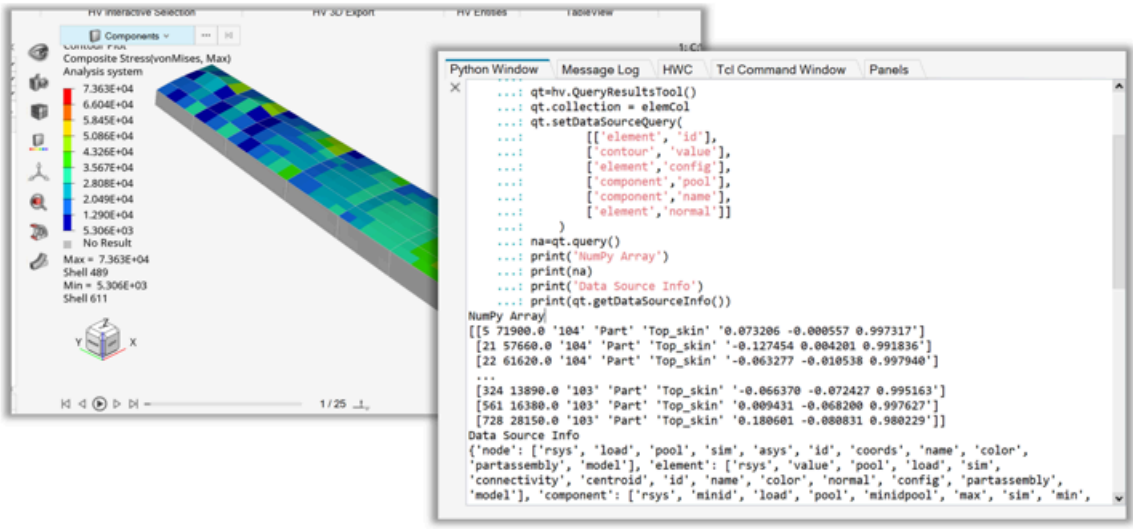

Figure 41:

- Using the newly introduced InteractiveSelection() tool allows you to create contexts for interactive selections:
  - Supported entity types are Part, Node, and Element
  - Return types are collections, lists and sorted lists of entity objects
  - Selection methods supported are single-, window- and advanced selections

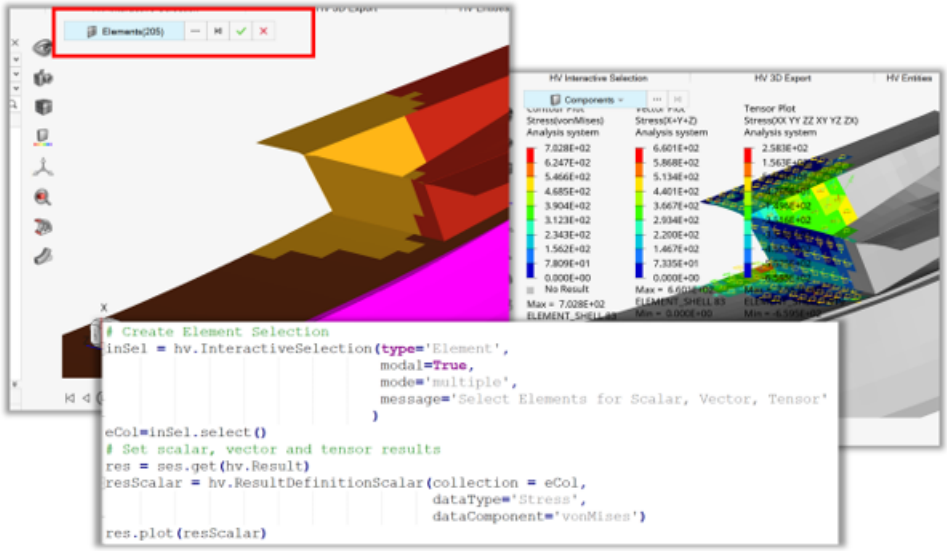

Figure 42:

- The new entity type class Set() supports all attributes of HyperView Sets. Entities can be added/removed by Collection and Sets using the methods addByCollection(), removeByCollection(), addBySet(), and removeBySet().

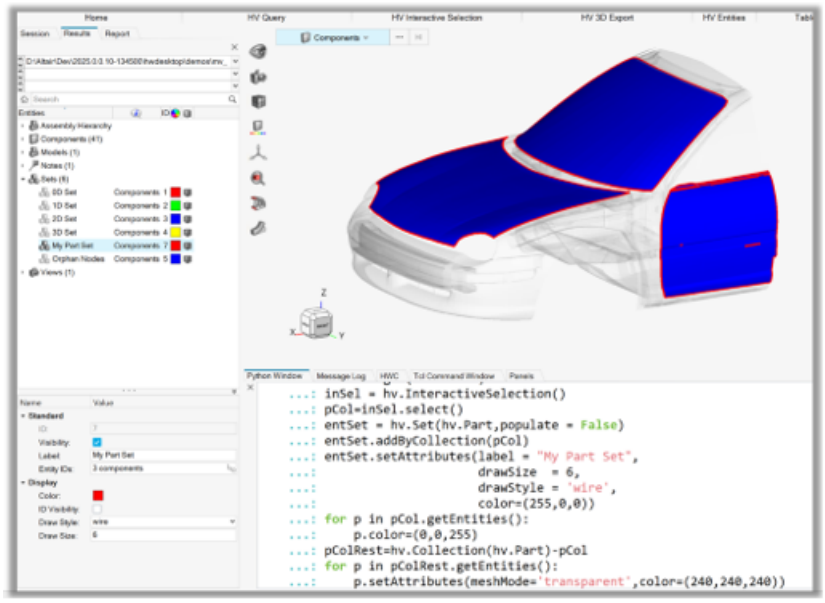

Figure 43:

- The Python classes ExportModelGLBinPPT(), ExportModelH3DinPPT(), ExportModelH3DinHTML(), and ExportModelGLB() support 3D Model Export from HyperView. All attributes from the Export 3D Model graphical user interface are supported.

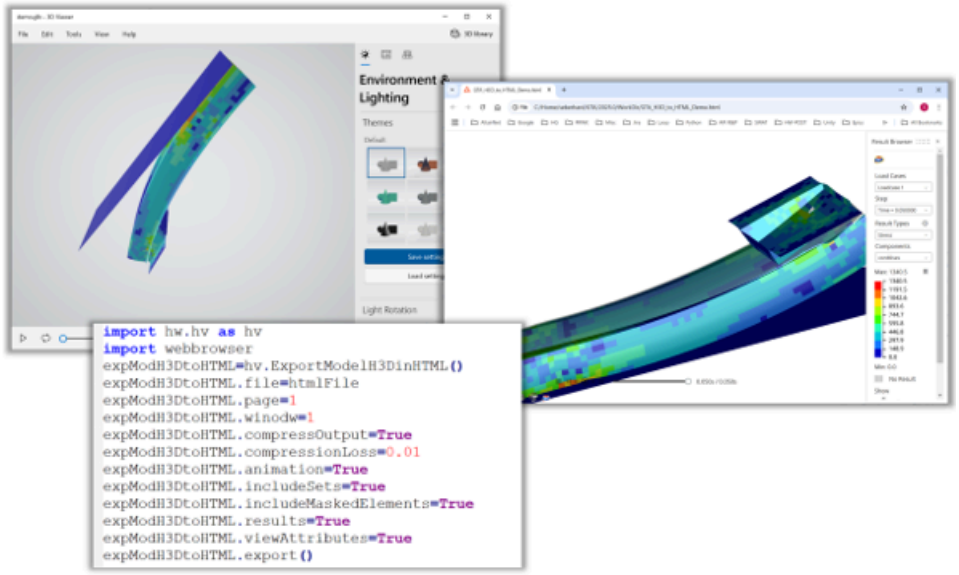

Figure 44:

- **Safety Tools: New Upper Interior Impact Post Tool**

  The Upper Interior Impact tool facilitates post-processing of the FMVSS201U analysis. It available for the Radioss, LSDyna and Pamcrash Interface profiles.

Figure 45:

This tool uses time-history and result files as input and automates the post-processing with minimal input, therefore reducing the report generation lead time.

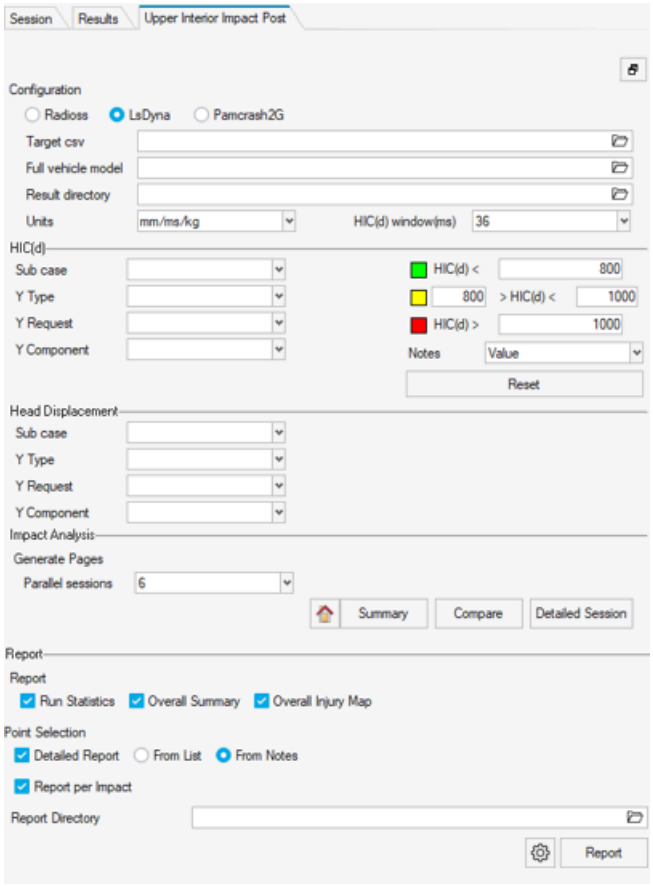

Figure 46:

For each Target point detailed, individual session pages will be shown for simulation results.

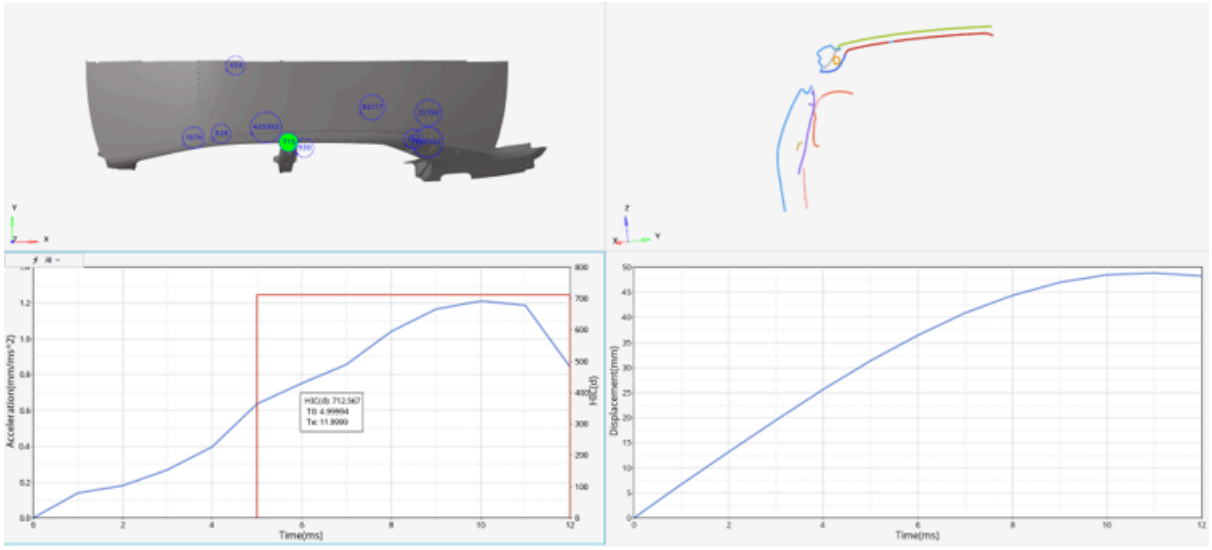

Figure 47:

Enhancements

- Terminology change: The names of some result-related options have changed in this release.
  - In HyperView - MultiCore, “simple” averaging method will now be referred to as “compute then average” and “advanced” method is now referred to as “average then compute”.
  - “Abs Max Principal” result component is renamed to “Extreme Principal” in both the HyperView - General and HyperView - MultiCore profiles.
- Max Abs Principal: This new computed result to calculate the absolute maximum of the three principal stresses or strains is available now in both HyperView profiles.
- Ctrl + MMB is updated to set rotation center only and auto-fit will be mapped to Alt + MMB.
- Notes default fill color is changed to make it more visible in light and dark themes.

Resolved Issues

- View Cube > Standard option not working properly.
- HyperView - MultiCore application hangs when loading certain h3d files.
- Selector button text is truncated in some UHD machines.
- Hotspot Finder tool created notes attached to incorrect entities in case of duplicate IDs.
- Select all or reverse of loadcases slow when dealing with results containing large number of loadcases.
- Application crash when editing a note on a model without result.
- Application error when turning on/off Axisymmetry from display control in HyperView Player.
- Shortcut keys are not working when selecting entities in Measures tool.
- HTML files did not show labels on View Cube.
- In HTML, Callout table did not save preferences of RMB.

Extensions

Enhancements

Improved Version Administration

When multiple extensions with the same name and author but different versions and minimum product versions are registered, the extension with the most fitting `"minProductVersion"` and with the highest `"version"` will be loaded first.

```
<entry name="minProductVersion" value="2024.0" />
<entry name="version"           value="2.0" />

<entry name="minProductVersion" value="2025.0" />
<entry name="version"           value="3.0" />

<entry name="minProductVersion" value="2025.0" />
<entry name="version"           value="4.0" />
```

Figure 48:

Improved Messaging

Improved messaging when an extension fails to register. More information about other registered extensions with the same name and author are now provided.

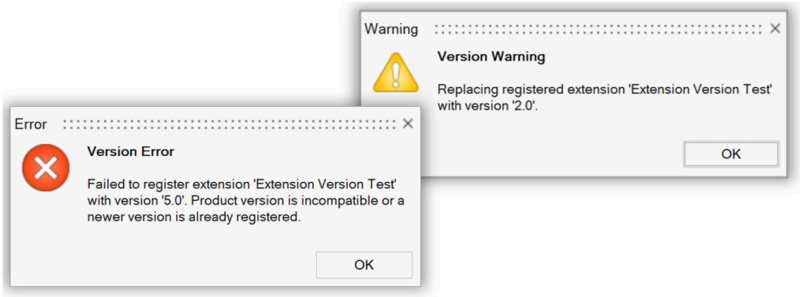

Figure 49:

$EXTENSION_DIR Variable Support

The `$EXTENSION_DIR` variable is now supported for all command attributes in the XML files. The previously limitation that the `$EXTENSION_DIR` variable was supported only for the command attribute no longer applies, all command type attributes are now supported.

```
<item tag="lineedit" type="combobox"
valuelistcommand="py: $EXTENSION_DIR.logic.comboValues()"
getcommand="py: $EXTENSION_DIR.logic.comboGet()"
setcommand="py: $EXTENSION_DIR.logic.comboSet()" />
```

Figure 50:

Altair HyperGraph 2025 Release Notes

General

New Features

Added Single Vector Macros to the Stackmath Workflow
The stackmath user interface has been replaced with the Math Macro dialog to allow for additional coverage.

Tip to Tail Tool
The Tip to Tail tool has been migrated from a panel to a dialog. From the Vector Chart ribbon, select the **Tip to Tail** tool.

Enhancements

Reader Enhancements
Support for the `.CUR` format for OASIS T/HIS software.

Resolved Issues

- XY data import.
- 3D zoom and pan.
- Nte calculation issue has been resolved for crash use cases.
- Secondary axis scaling when units are activated.

Extensions

Enhancements

Improved Version Administration
When multiple extensions with the same name and author but different versions and minimum product versions are registered, the extension with the most fitting `"minProductVersion"` and with the highest `"version"` will be loaded first.

```
<entry name="minProductVersion" value="2024.0" />
<entry name="version"           value="2.0" />

<entry name="minProductVersion" value="2025.0" />
<entry name="version"           value="3.0" />

<entry name="minProductVersion" value="2025.0" />
<entry name="version"           value="4.0" />
```

Figure 51:

Improved Messaging

Improved messaging when an extension fails to register. More information about other registered extensions with the same name and author are now provided.

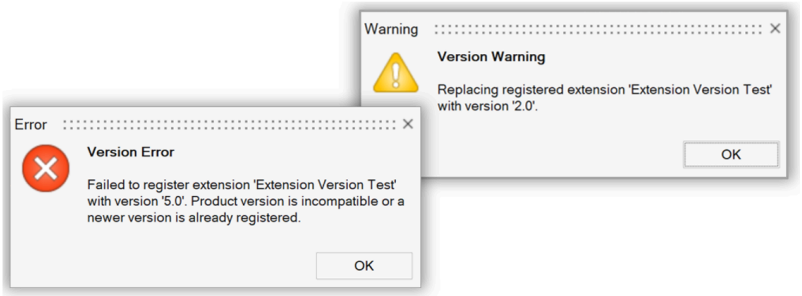

Figure 52:

$EXTENSION_DIR Variable Support

The `$EXTENSION_DIR` variable is now supported for all command attributes in the XML files. The previously limitation that the `$EXTENSION_DIR` variable was supported only for the command attribute no longer applies, all command type attributes are now supported.

```
<item tag="lineedit" type="combobox"
valuelistcommand="py: $EXTENSION_DIR.logic.comboValues()"
getcommand="py: $EXTENSION_DIR.logic.comboGet()"
setcommand="py: $EXTENSION_DIR.logic.comboSet()" />
```

Figure 53:

Altair MotionView 2025 Release Notes

Altair Multibody Solutions 2025 Release Notes

Highlights

The 2025 release of Altair Multibody Solutions introduces several enhancements and bug fixes to MotionView, MotionSolve, and their extensions. Here are some of the key highlights:

- Save system definition to the Entity Browser (MotionView)
- Deformable curve on a Polybeam (MotionView)
- Improvements to Frequency Response Analysis (MotionSolve)
- CD Tire updates (MotionSolve/Vehicle Tools Extension)

New Features

Save System Definition to the Entity Browser (MotionView)

MotionView's Entity Browser contains a list of entities that can be added to the model. It is now possible to save systems to the Entity Browser for quick retrieval and reuse.

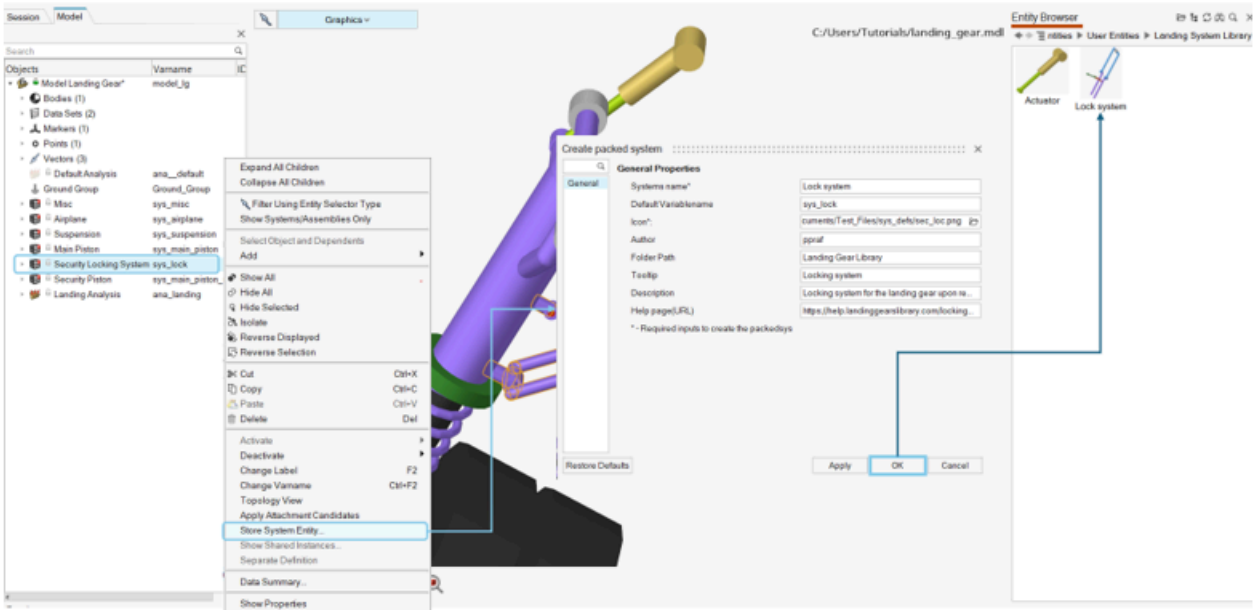

Figure 54:

Select a system in the Model Browser, right-click, and select **Store System Entity...**. In the subsequent dialog, fill in system details, an icon image, and click **OK**.

- The system is listed under the User Entities folder.
- Provide a folder path to organize the stored systems.
- Double click on the stored system to insert it into the active model.

Deformable Curve on Polybeam (MotionView)

In the previous release, MotionView and MotionSolve added support for the deformable curve contact, which can be used to model pipes, cables, and tubes in contact. In this release, MotionView further enhances this functionality by providing an option to add a deformable curve with graphics when creating a polybeam entity. This improvement streamlines the setup of models with slender elements, enabling faster configuration.

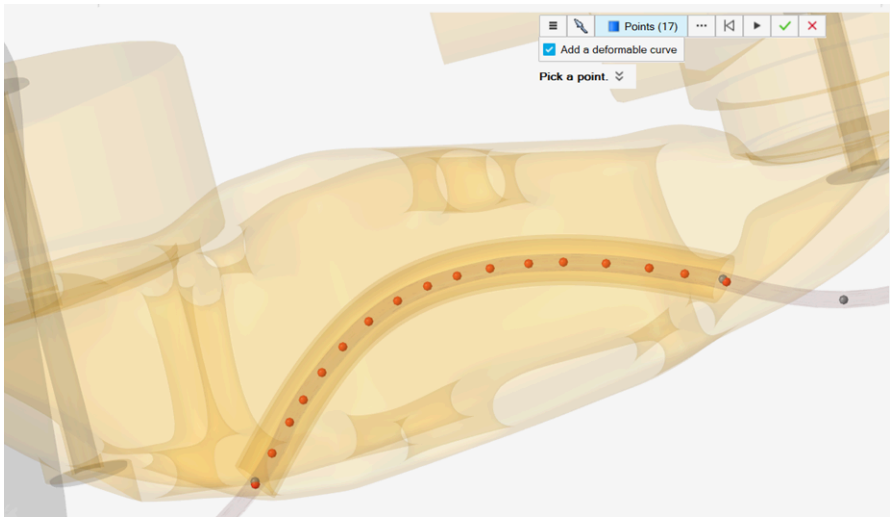

Figure 55:

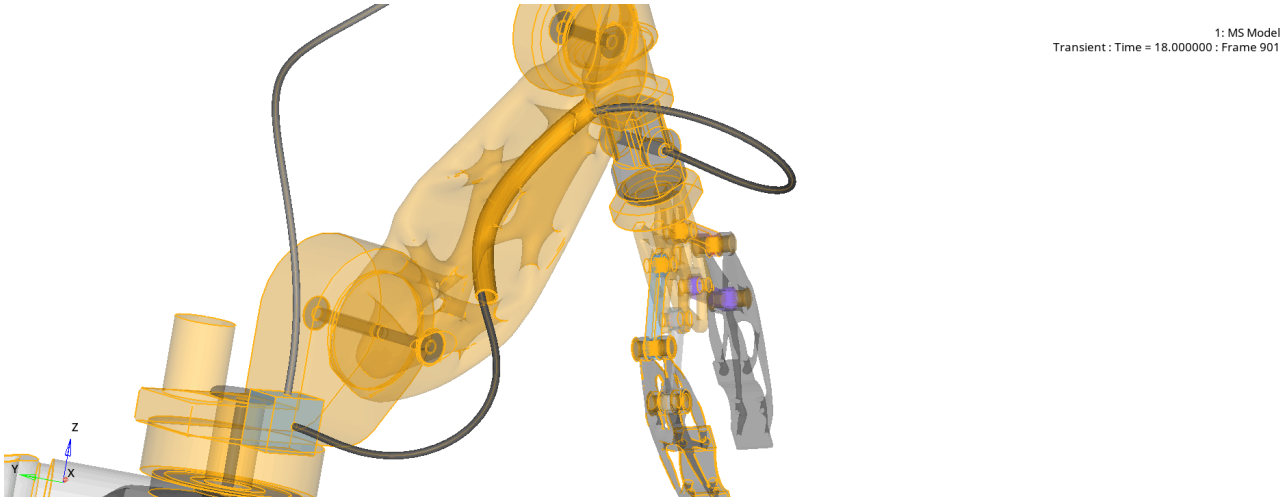

Figure 56:

Enhancements

Import Markers and Geometrical Curves from Catia (MotionView)

The CAD import functionality has been enhanced to read any existing marker and geometrical curve from Catia V5.

Improvements to Frequency Response Analysis (MotionSolve)

In this release, the Frequency Response Analysis in MotionSolve introduces additional support for flexible bodies:

- Frequency-dependent modal participation factors.
- Frequency-dependent kinetic energy, dissipative energy, and strain energy distribution.
- Frequency-dependent transfer path analysis.

Write mkb Matrix in Linear Analysis (MotionSolve)
In this release, MotionSolve includes support for exporting of the mass, stiffness, and damping matrices of a linearized system.

Fraunhofer CD-Tire: Resizing (Vehicle Tools Extension)
The Vehicle Tools Extension now supports the Resizing feature for CDTire from Fraunhofer ITWM. The resizing feature allows a tire model to be automatically resized based on reference and target tire and rim specifications.

Fraunhofer CD-Tire: Support for MF++ (Vehicle Tools Extension)
The Vehicle Tools Extension now supports the Fraunhofer MF++ tires. CDTire/MF++ is a temperature enhanced Magic Formula for coupling to CDTire/Thermal in advanced handling applications.

Fraunhofer CD-Tire: Licensing Changes (MotionSolve)
MotionSolve has updated the license check-out logic for CDTire and the CDTire resizing feature. MotionSolve supports the usage of the CD-Tire resizing feature. To use this feature, your Altair license must include both the CDTire_MSolve (regular CD Tire) and CDTire_MSolve_RS (CD Tire with resizing) features. MotionSolve first checks out the CDTire_MSolve feature, followed by the CDTire_MSolve_RS, if your model demands tire resizing.

Version Up for Siemens MF-Tyre/MF-Swift and Cosin FTire (Vehicle Tools Extension)
The Vehicle Tools Extension has upgraded the support for Siemens MF-Tyre/MF-Swift to v2406 and Cosin FTire to 2024-3. These new versions are backward compatible with tire and road input files from previous versions.

Motion-ratio Outputs for Car/Small Truck and Two-Wheeler Library Models (Vehicle Tools Extension)
The Vehicle Tools Extension includes an inbuilt output for reporting motion-ratios of ride springs for the models that are built using the Car/Small truck and Two-wheeler libraries. The calculations are based on the spring displacements and the corresponding wheel center displacements. When you build the models using the libraries, these outputs will be created in dedicated Systems alongside the springs whose motion-ratios are measured.

*Added Contact Jacobian Computation Time to *.`cof` File (MotionSolve)*
When requested, MotionSolve outputs a *.`cof` file summarizing the contact simulation. For each contact, it provides contact details, including the time spent on contact evaluation. In this release, the *.`cof` file is further enhanced to include the computation time for the contact Jacobian as well.

MODE and GETMOD Returns 35 for Assembly Analysis (MotionSolve)
The solver functions MODE and GETMODE provide insight into the current analysis mode of the solver. Both functions have been enhanced so that MotionSolve now returns a value of 35 during Assembly analysis, allowing for clearer differentiation between Assembly and Static analysis modes.

Resolved Issues

- Run time error is encountered after activating joint compliance.
- The **File** > **Open** dialog goes behind the Leaf Spring Builder tool in Linux.

- Output on Entity Sets within a system results in requests for all entities in the model.
- Constraint Mate is not supported with msolve Python.
- The Advanced Selection dialog cannot be invoked from collectors within the FrequencyInput Property Editor.
- Adding a Bushing attachment to a system results in an application error.
- The application crashes when solver variables are added to solver arrays in Linux.
- Entities lose their labels in the Model Browser after undoing the paste action in a system.
- Creating a second Belt pulley system (Substructured flexbodies) takes too long.
- mdl_batch exports an incomplete model from the Vehicle Tools in Linux.
- A force created on a marker has a missing reference after creation.
- The topology view on systems is unreadable when it contains a non-connected entity.
- FMU exported from MotionView is not readable in HyperStudy.
- Y and Z axes in Spline 3D are swapped in the display.
- Zero crossing sensor hmax related to contact is always written despite its selection.
- Updated file paths in FMU parameters are not updated in the exported XML.
- The Contact Force plot is missing in the default Contact report.
- Opening two optimization models consecutively crashes MotionView.
- Datasets with the same variable name have the values synchronized.
- Initial Conditions on systems input through the Entity Editor are not applied to the model.
- Numerical Jacobian calculation for flexible body.
- Functions QDOT and QDDOT must use an actual mode number instead of an index.
- Crash when writing the load set file when the flexible body is not used in any contact.
- Differences in EDEM co-simulation results: Difference between starting the coupling server in Creator Mode and Simulator Mode.
- `edem.fmu` randomly fails to load during the simulation start.
- Model with UserGra causes CNAME to display on all requests in the ABF file.
- mspost now displays the correct progress during ABF export.
- MotionSolve continues saving the XML even if the *savesub* failed.
- MotionSolve supports both label and comment for *post_request* in XML.
- MotionSolve writes only the converged results at the start time of a kinematic analysis to MRF.
- Models with softsoil road graphics created using RoadTools in v2024.0 and v2024.1 are failing.
- CDTire models fail when only one license for CDTire Resizing is available.

  Note: The licensing logic has been changed. Please see *Enhancements* for more details.

- Anti-dive calculations do not report correct values.
- Models with CDTire and FTire crash the solver if the initial static is not included.
- Two-wheeler models with a leaning driver fail after traveling a certain distance for specific event parameters, especially in a Constant radius cornering event.

Known Issues

The following known issues will be addressed in a future release as we continuously improve the software:

- Accessing custom wizards crashes MotionView.

Altair MediaView 2025 Release Notes

Enhancements

- Updated the MediaView browser icons for light and dark themes.

Altair TableView 2025 Release Notes

New Features

Overlay Tool

The Overlay tool has been migrated from a panel to a dialog. From the Table ribbon, select the **Overlay** tool.

Python API for TableView

Complete Python API coverage for TableView has been added in this release. This includes the Window() class of type 'table':

- CSV import and export.
- Setting multiple cell values via a dictionary.
- Formatting cell fonts, text, and background colors.
- Dynamically adapting the table size relative to the table content.

Use the classes below to add the following features:

- TableRule() class to add conditional formatting.
- TableOverlay() classes to overlay areas for report templates.

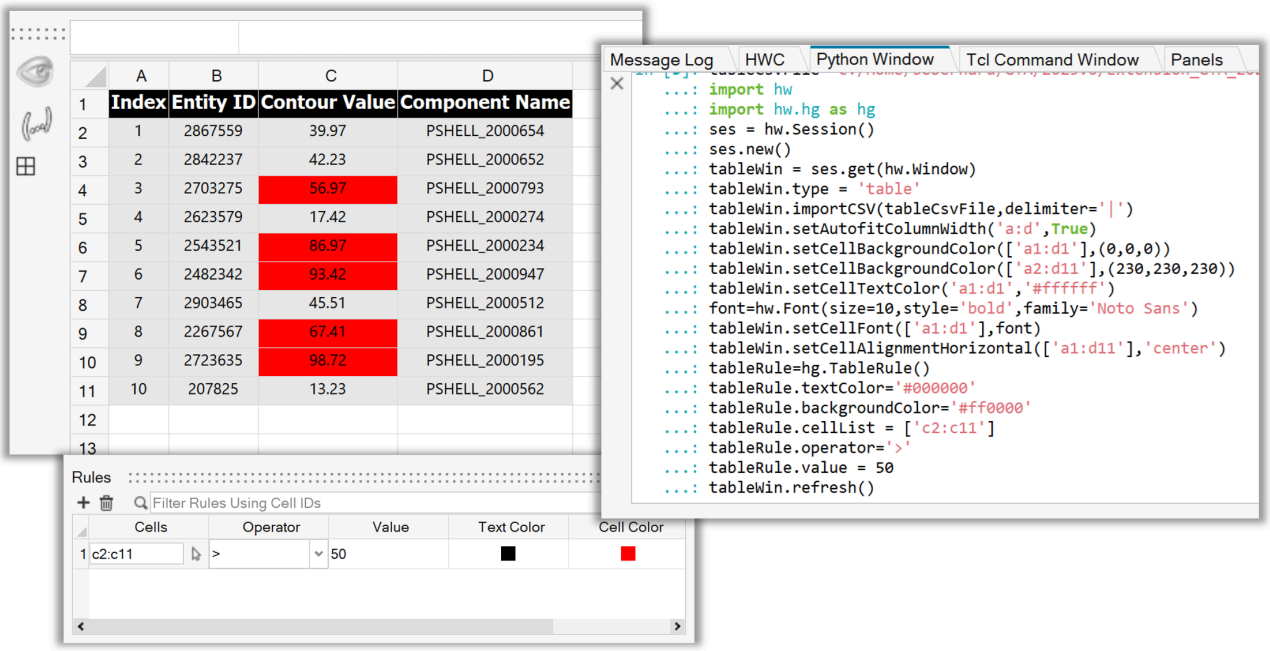

Figure 57:

Altair TextView 2025 Release Notes

No updates to TextView for this release.

Extensions 2025 Release Notes

Enhancements

Improved Version Administration

When multiple extensions with the same name and author but different versions and minimum product versions are registered, the extension with the most fitting `"minProductVersion"` and with the highest `"version"` will be loaded first.

```
<entry name="minProductVersion" value="2024.0" />
<entry name="version"           value="2.0" />

<entry name="minProductVersion" value="2025.0" />
<entry name="version"           value="3.0" />

<entry name="minProductVersion" value="2025.0" />
<entry name="version"           value="4.0" />
```

Figure 58:

Improved Messaging

Improved messaging when an extension fails to register. More information about other registered extensions with the same name and author are now provided.

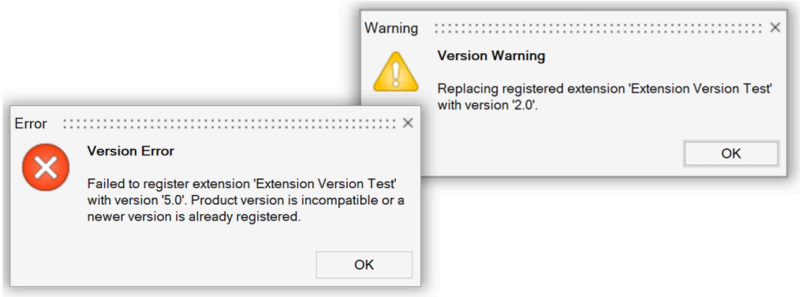

Figure 59:

$EXTENSION_DIR Variable Support

The `$EXTENSION_DIR` variable is now supported for all command attributes in the XML files. The previously limitation that the `$EXTENSION_DIR` variable was supported only for the command attribute no longer applies, all command type attributes are now supported.

```
<item tag="lineedit" type="combobox"
valuelistcommand="py: $EXTENSION_DIR.logic.comboValues()"
getcommand="py: $EXTENSION_DIR.logic.comboGet()"
setcommand="py: $EXTENSION_DIR.logic.comboSet()" />
```

Figure 60:

Altair PhysicsAI 2025 Release Notes

Announcements

License checkout for prediction outside Altair GUI products, like HyperMesh or Inspire, will now draw 150 Altair Units.

New Features

New Architecture Introduced - Transformer Neural Simulator (TNS)

PhysicsAI now has a new AI training method - the Transformer Neural Simulator (TNS). This is in addition to the existing method, the Graph Context Neural Simulator (GCNS). Some key differences between the two:

- Usually, TNS should predict smoother contours than GCNS.
- TNS is less sensitive to variation in mesh sizes.
- On a GPU, TNS is faster than GCNS while on a CPU, it is typically the opposite.

Natively Reading Simulation Entities (Thickness and Material IDs)

PhysicsAI can now extract thicknesses and material IDs if there are solver decks available along with the training files. These decks should be in the same location and have the same name as the corresponding solver file. For example, if a file called `Ibeam.h3d` has a file accompanying `Ibeam.fem`, then the thicknesses and material IDs can be parsed by selecting **Extract Simulation Properties**.

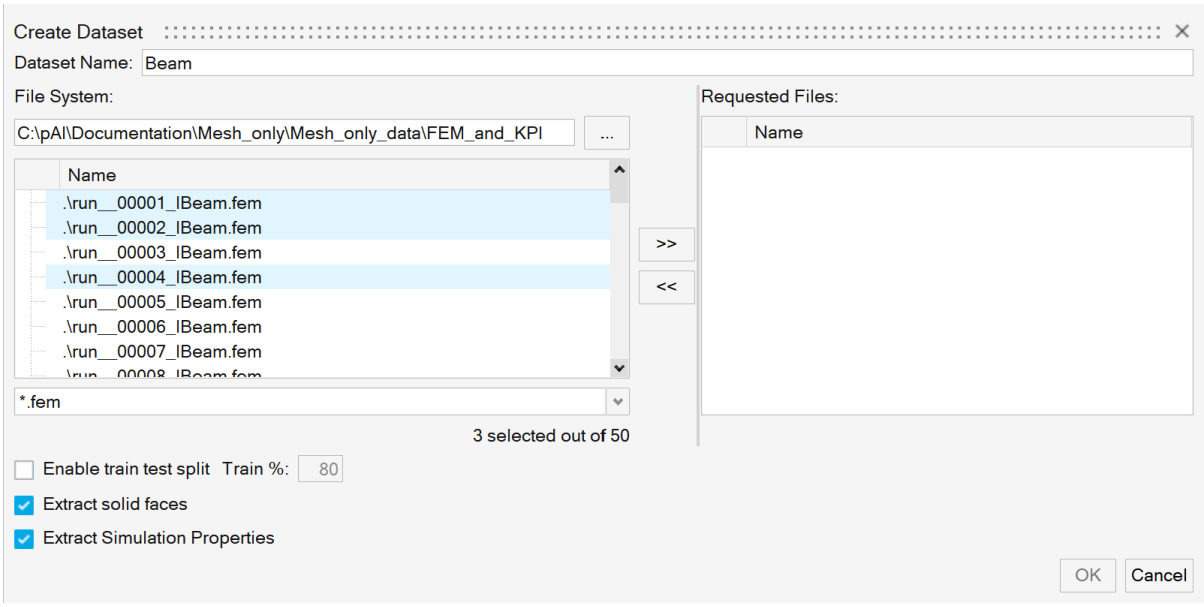

Figure 61:

The list of supported solver decks include: Optistruct, Radioss, LS-DYNA, Nastran, Abaqus, and ANSYS.

Enhancements

Mesh Alignment for Translational Invariance

PhysicsAI is sensitive to meshes being translated and rotated in space. Earlier, you had to manually orient the meshes to eliminate this source of noise. Now, using the Mesh Alignment feature during model training, the meshes can be adjusted during training, testing, and prediction. The meshes are aligned such that the center of gravity is coincident at a common point. This option can only correct translational variances and not rotational variances.

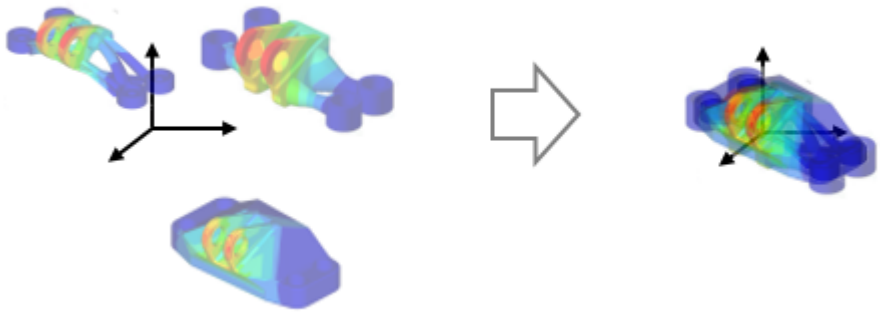

Figure 62:

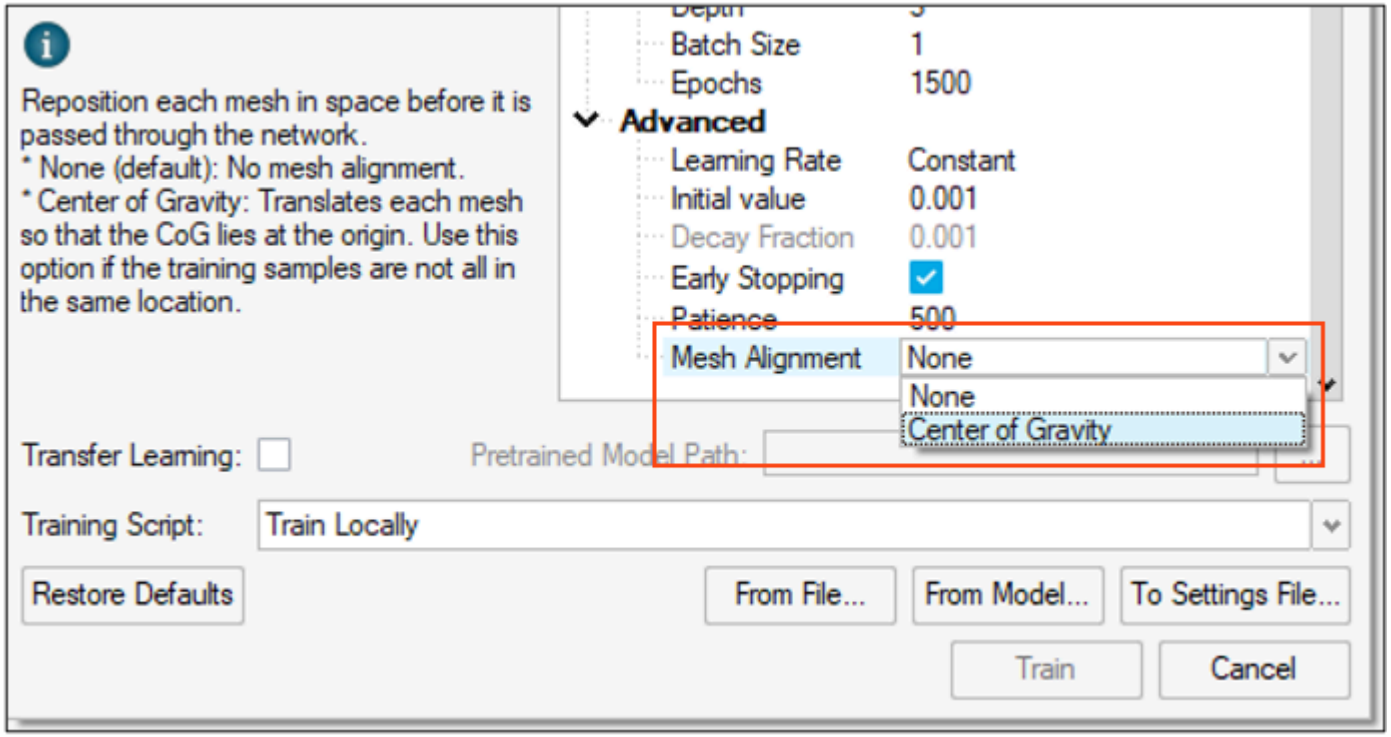

Figure 63:

Enhanced Dataset Visualization and Outlier Detection

Previously, you had to utilize other tools, such as HyperStudy, to curate the data and identify outliers. Now, outliers in the dataset are identified based on the you select. A Z-distribution is fitted to the data and points that fall in the 3-sigma tails are highlighted as outliers.

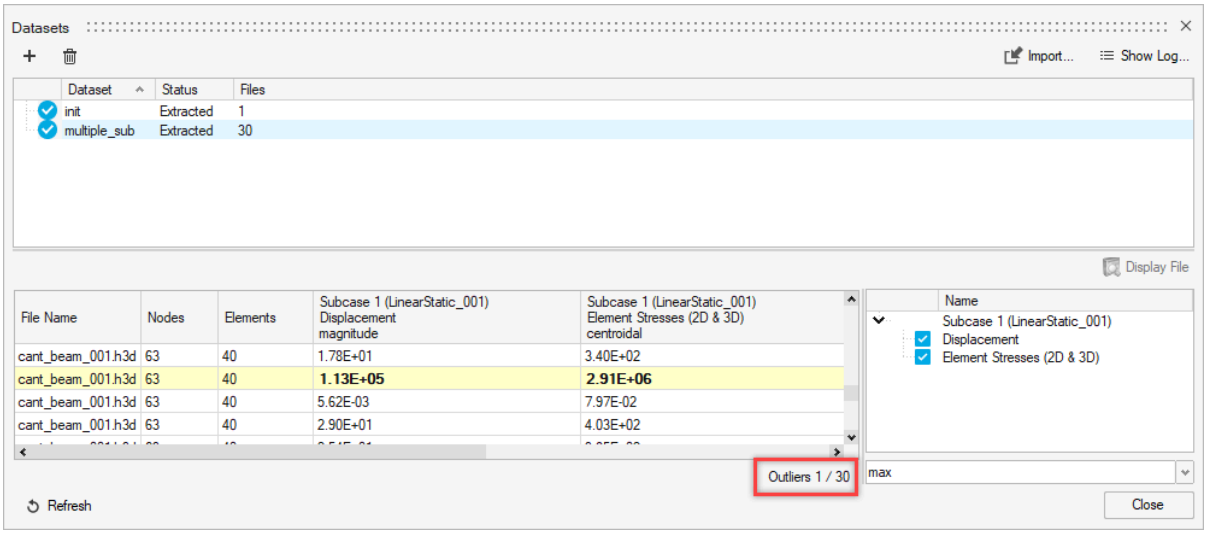

Figure 64:

Similarity Score in the PhysicsAI Connection in HyperStudy

You can now access the Similarity score for a prediction made using a physicsAI model in HyperStudy. This can be a useful response to qualify results based on expected accuracy. For example, using it as a constraint to reject predictions with low Similarity scores. The Similarity score is automatically added as a response if it is available in the selected physicsAI model.

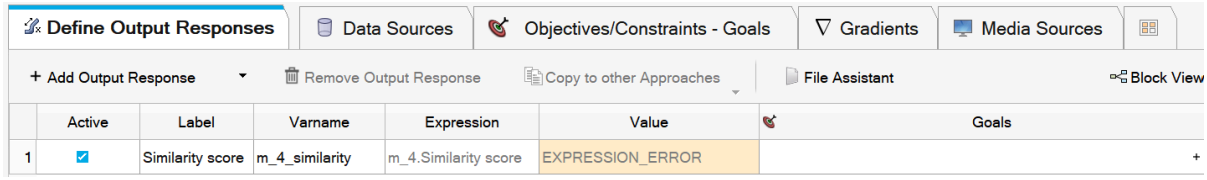

Figure 65:

Known Issues

- Predictions made in the Radioss solver profile within HyperMesh may not function correctly when model features are used (like thickness or material). This issue can be avoided by using another solver profile, such as OptiStruct.

Resolved Issues

- Eroded elements are now hidden during visualization. Previously, eroded elements resulted in an exploded mesh resulting in confusing visualization. Now, elements are hidden after the point of erosion.

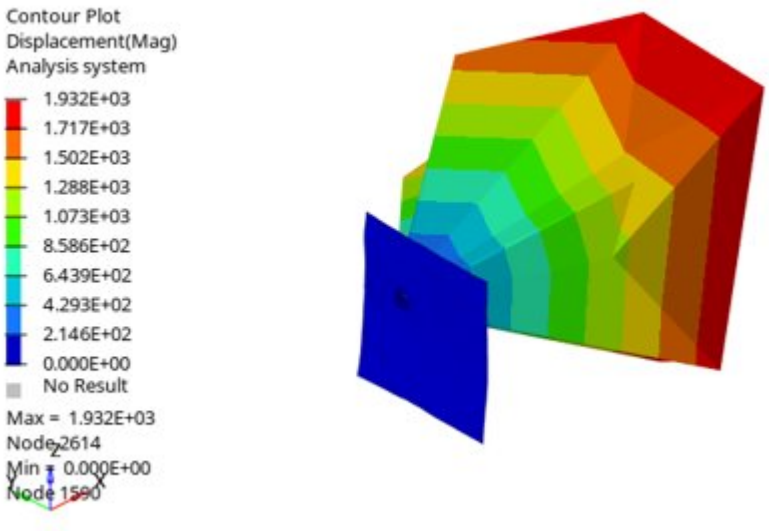
Contour Plot
Displacement(Mag)
Analysis system
1.932E+03
1.717E+03
1.502E+03
1.288E+03
1.073E+03
8.586E+02
6.439E+02
4.293E+02
2.146E+02
0.000E+00
No Result
Max = 1.932E+03
Node 2614
Min = 0.000E+00
Node 1590
Z
Y
X

Figure 66:

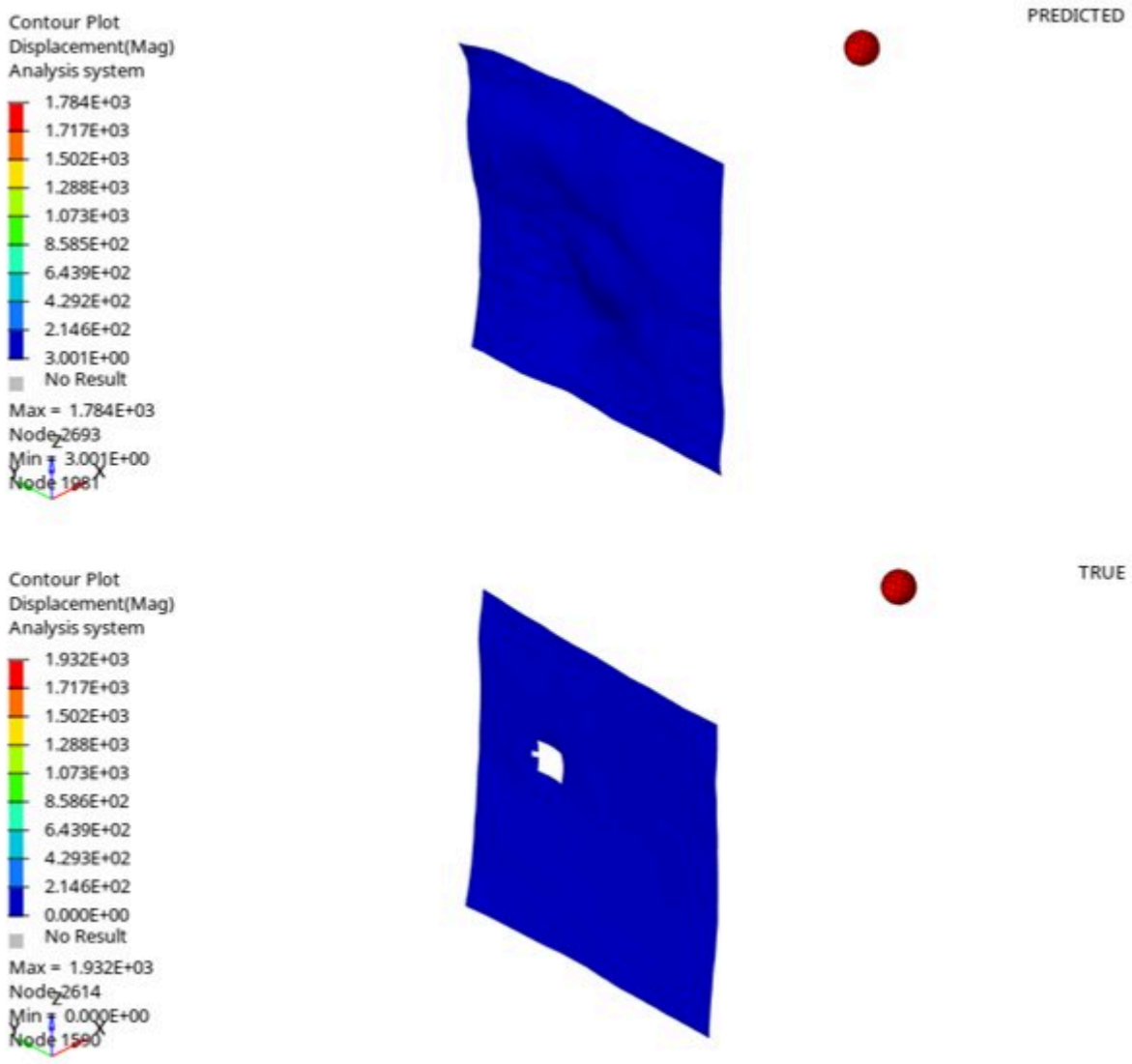

Figure 67:

- Previously, errors were encountered if the training data contained parts with no results. Now, physicsAI automatically excludes such parts and the training can continue.

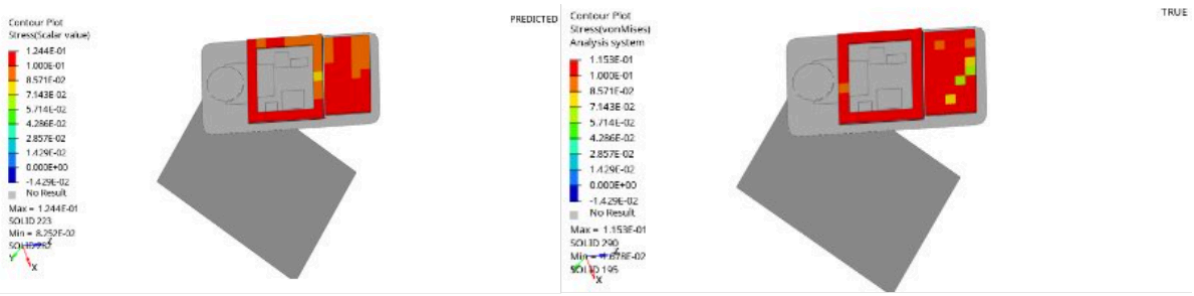

Figure 68:

Aerospace 2025 Release Notes

No updates were made to Aerospace in 2025.

NVH Director 2025 Release Notes

Highlights

- MAC Optimization – Support for Abaqus input and output files.

New Features

MAC Optimization - Support for Abaqus input and output files

New feature to support MAC optimization for Abaqus input (`.inp`) and output (`.odb`) files. This will allow you to execute MAC optimization for Abaqus files.

Enhancements

External Cavity Mesher

New options for ellipsoid and cuboid for near field mesh.

CDTire

Enhancement to show user message for the minimum patch width and PreLoad after Linearization.
Enhancement to turn on/off the cavity flag before linearization.
Enhancement to show and edit the number of cross-sections.

FRAC

Enhancements to improve the performance.

Resolved Issues

- Issue related to Spindle Forces calculation in case of mismatch in the frequency content between transfer function results and acceleration PSD results.

Altair Squeak and Rattle Director 2025 Release Notes

New Features

Variability Study in Batch Mode

A new feature has been added to export the variability study to run in batch mode locally.

Automatic Reporting

A new feature has been added to create an automatic summary report for the squeak and rattle assessment.

HyperWorks Report 2025 Release Notes

New Features

Python API

Documents can now be created using Python APIs. The newly added classes are:

- Document()
- Chapter()
- DocText()
- DocImage()
- DocTable()

DocText(), DocImage(), and DocTable() define a text/image/table inside the document.

They can be constructed using one of the following:

- A Text/Image/Table object.
- A `string/image` or `path/csv` file path, respectively.

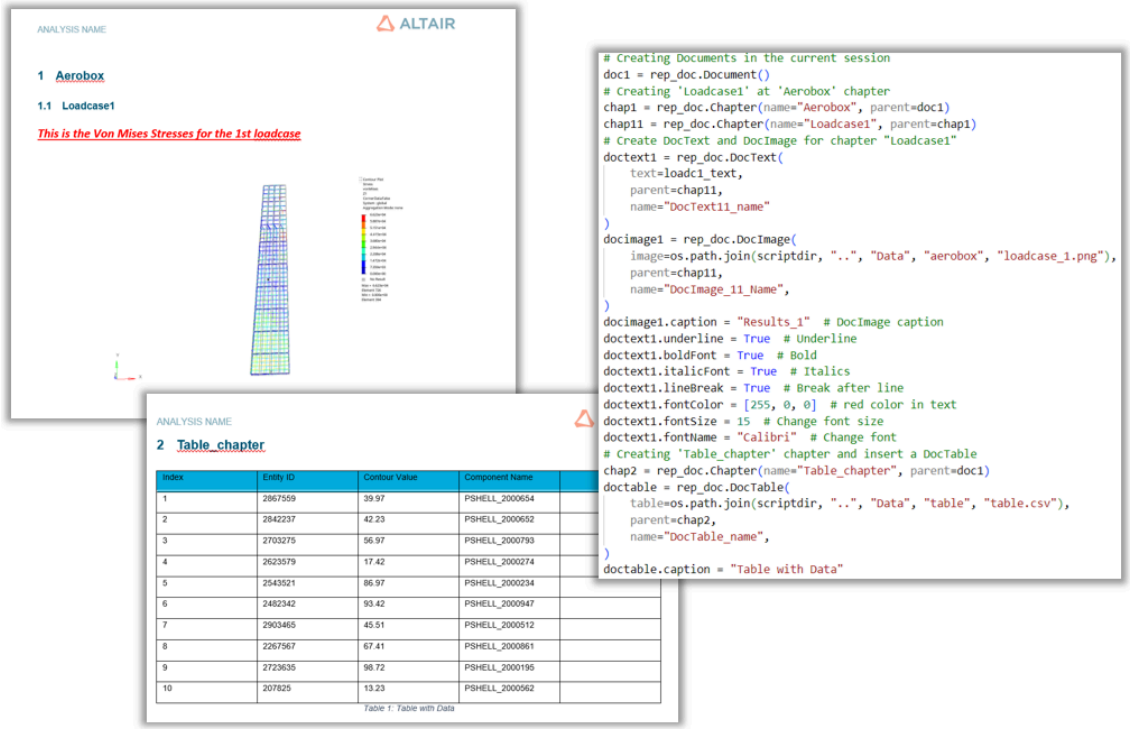

Figure 69:

Use the Document() class to provide a *.`docx` template and export the document.

```
# Set a template and export
doc1.template = rf"{hm.altair_home}/mv/scripts/tcl/report/templates/wordTemplate.docx"
doc1.saveDocx(
    os.path.join(scriptdir, "..", "Export", f"{doc1.name}.docx"),
    open=True,
    indexpages=True,
)
```

Figure 70:

Inside the document, the *parent* attribute defines a parent-child relationship in the content. You can also use this to change the content's position, or create sub-chapters within chapters.
For the Document() and Chapter() classes, the getChildren() method provides a list of the related child objects.

```
In [16]: chap1.getChildren(type=[rep_doc.DocImage], childrenLevel="all")
Out[16]:
[<report.hwxdocxtypes.DocImage at 0x21993f12a00>,
 <report.hwxdocxtypes.DocImage at 0x21993f5b8b0>]
```

Figure 71:

Enhancements

Python API

In the ReportSession() class, use the paste() method to paste content from one Document/ Presentation or Chapter/Slide to another.

```
# Paste all DocImages of chapter1 to chapter2
session.paste(chap1.getChildren(type=[rep_doc.DocImage],childrenLevel="all"),chap2)
```

Figure 72:

In the Image() class, the page and window properties have been added. Use these properties to set the ID of the page or window that you want to capture a GUI image from.

Resolved Issues

- Slide layout selection was missing for the module Capture Pages and Windows.

CAD and Solver Interfaces

Abaqus Interface 2025 Release Notes

Altair HyperMesh

Enhancements

LARC05 Damage Model
A new LARC05 damage model has been added for composites.

STRUCTURAL Option Support
For the *DAMPING keyword, the STRUCTURAL option is now supported for Abaqus/Explicit profile.

Resolved Issues

- The SPH tool is now available in search for Abaqus/Explicit profile.
- Nodes with system assignment are no longer lost on import.
- An issue with reading *SHELL SECTION and *SHELL SECTION, COMPOSITE has been resolved.
- A warning message has been added if an Abaqus parts and instances deck is exported in flat format.
- A solver keyword for *COUPLING and *KINEMATIC/*DISTRIBUTING has been added to the Entity Editor.

ANSYS Interface 2025 Release Notes

Altair HyperMesh

New Features

The TB(JOIN) card is now supported for stiffness, friction, and damping.

CAD Interface 2025 Release Notes

Altair HyperMesh

New Features

- NX 2406 support has been added (Native reader).

- 2024 3DXML support has been added.
- Step AP242 XML is now supported for BOM Export.

Enhancements

- Inspire Parameter update is now supported on Linux.
- The CAD Reader log file extension has been changed from `*.msg` to `*.log`.

OptiStruct Interface 2025 Release Notes

Altair HyperMesh

New Features

Encryption Functionality

Encryption functionality is now available in OptiStruct input files to encrypt confidential information.

- Properties and materials can be imported and exported with encryption.
- Damage and table cards associated with property and material entities can also be encrypted.
- Additionally, if the supported entities are defined within an INCLUDE file, the entire INCLUDE file can be encrypted.

Enhancements

Mass Calculation of NSM

Mass calculation of NSM defined through SETs with property.

Known Issues

- SUBCASE cannot be selected for ERP constraint.
- NU option cannot be cleared once populated on DREPORT card.

PAM-CRASH 2G Interface 2025 Release Notes

Altair HyperMesh

Enhancements

PAM-CRASH 2024 Profile Support

Support of the PAM-CRASH 2024 profile has been added.

Radioss Interface 2025 Release Notes

Altair HyperMesh

New Features

- A new Initial State entity has been introduced and supports all the following keywords: /INISHE, /INISH3N/, /INIBRI, and so on.

Enhancements

- A new Model Checker check has been added for RBODY in component with Part card image instances with auto correction.
- Mass calculation has been improved in the Validate and Mass Measurement workflows when a lumped mass is applied on a free RBODY secondary node.

Known Issues

- After switching from the OptiStruct to Radioss solver profile, the RBODY node set is now correctly created on export.
- Auto correction of the Model Checker check for RBODY main node on element instances now works for two nodes RBODY type.

Altair HyperForm 2025 Release Notes

Enhancements

STA file size reduction from Radioss incremental solver run

The STA file exported from the solver run is broken into a smaller mesh file and compressed file (`str.gz`) that has all the initial conditions to reduce memory footprint. Use the STA file as usual to set up multistage analysis. Keep the `str.gz` file from the previous stage in the run folder.

Results initializer

The following additional parameters can now be defined for each part:

- Draw direction
- Interface friction

A summary report in html format is now available with the following columns when only feasibility analysis is picked:

- Part name, ID, thickness, material, blank holding force condition, maximum thinning, maximum plastic strain

Resolved Issues

- Tube bending and hydro bending can now handle tube parts with special characters in their names.

Altair HyperLife 2025 Release Notes

Highlights

- Improvement to Stress Life evaluation to support scale factor output to achieve the required life for the given stress history.
- Improvement to Strain Life evaluation to support Standard Error of Cyclic Stress-Strain (SEc).

New Features

Stress Life (SN) evaluation to Safety Factor = Scale
In addition to the damage calculation, HyperLife evaluates the scale required to obtain a target damage or life, when safety factor is requested with option **Scale**.

Strain Life (eN) evaluation supports Standard Error of Cyclic Stress-Strain
The Standard Error of Cyclic Stress-Strain (SEc) is defined via the Strain Life material properties. The SEc value is calculated by modifying the cyclic strength coefficient.

Enhancements

Launch Log file from Run Status
HyperLife Run summary post evaluation can be directly launched from the **Run Status** dialog, by clicking the run name.

Resolved Issues

- Run name is now retained on reopening the Evaluate context and on reloading the session file.
- HyperLife preferences now reverts to default for number of processes.
- Empty `H5` file written no longer when no rainflow output is requested.
- Assigning material to entity sets no longer shows application error.
- The number of processors from **File** > **Preferences** is now reflected in the **Evaluate** > **No of processors**.

Known Issues

- Material selection is not retained for the spot weld after weld fatigue analysis. The issue is inconsistent and will be fixed in the upcoming release.
- Layer should be manually selected for *PERMAS H3D* result file format for fatigue evaluation.

Altair HyperLife Weld Certification 2025 Release Notes

Highlights

There are no significant changes introduced in HyperLife Weld Certification 2025.

Resolved Issues

- The issue experienced when changing the active weld type has been resolved.

Known Issues

The following issues will be addressed in the upcoming release:

- Session file (`.hlf`) is currently not retaining the derived loadcases
- Application error while evaluating a session file (`.hlf`) from stress life analysis.
- Prior to loading the Abaqus results ODB file into a HyperLife Weld Certification session, upgrade it using the AbaqusODB Upgrade tool.

Altair HyperLife Crack Growth 2025 Release Notes

Highlights

- Updates to crack growth evaluation via batch evaluation and event group support.
- Event contribution XML is now output for additional results in post-processing.

New Features

Batch evaluation Support
: Support for HyperLife Crack Growth evaluation in batch. The run setup XML housing the hotspot XML is utilized as batch input for evaluation. Batch evaluation is supported on both Windows and Linux.

Support of Nested Events
: HyperLife LoadMap introduces “Event Groups” for the Crack Growth profile to define combination of events within events. Repeats can be applied to the events and the event groups to obtain the required sequence of loading in the duty cycle. Event Groups support is also extended to HyperLife LoadMap XML.

Support of Event Contribution
: For the duty cycle assigned to the crack growth evaluation, the percentage contribution of each event to the crack growth history are output as results to the run directory. An event summary CSV is output for each evaluated hotspot if more than one hotspot is chosen for evaluation.

Support of Min and Max Stress in the Duty Cycle
: The maximum and minimum stress from the entire stress history of the duty cycle is output via a CSV, post crack growth evaluation.

Resolved Issues

- An application error when trying to close a load history plot has been resolved.

Known Issues

The following issues will be addressed in the upcoming release:

- Right click menu options **Apply value to all events** and **Apply value to current event** from LoadMap event creation are not working as intended; the issue will be fixed in the upcoming release.

Altair HyperStudy 2025 Release Notes

New Features

Leverage accuracy metrics as responses

HyperStudy Pyfit model now provides an option to include fit metrics as conventional responses. This allows monitoring accuracy and applying objectives which is useful in fit-based optimization approaches.

Deploy PhysicsAI model as solver

A new extension tool to auto-register trained PhysicsAI models as solvers.

Enhancements

- Perform trade-off studies following input variable formats
- Tutorial HS-1100, Setting Up Existing Data Model
- Tutorial HS-1695, Using Hooks in PhysicsAI Model

Resolved Issues

- Using labels instead of model parameters in `hst_output.hstp` files no longer prevents extracting values of different outputs with same labels.
- Hidden or shown tabs no longer restore to default after reopening the application.

Announcements

- Data Source tool, Hstp Reader, has been deprecated and the updater will convert it to XML reader tool. Please use deprecate flag with 8319 to restore the tool.
- "*setDirectory*" and "*setFileHint*" python functions have been removed from External Optimizer API. They have been replaced with `getResourcePath(type: RESOURCE_TYPE)-> str` on the setup class using the resource types `RESOURCE_TYPE_WORK_DIRECTORY` and `RESOURCE_TYPE_FILE_HINT`.

Altair Mechanical Solvers 2025 Release Notes

Altair Compute Console 2025 Release Notes

- No updates to Altair Compute Console for this release.

Altair OptiStruct 2025 Release Notes

Highlights

- Global-local analysis support for random response analysis
- Curved beam or curved pipe element modeling with `CBEND`
- Supersonic aeroelasticity
- Implicit explicit chaining (beta)
- SPL optimization
- Electrostatic Analysis
- Symmetry constraints for topology and free-size in MMO

New Features

Stiffness, Strength, and Stability

Enhancement to `OUTPUT,MASSPROP` and `OUTPUT,MASSCOMP`
A new moment of inertia table is printed in the `.out` file which prints the inertia with respect to the basic coordinate system. The preexisting table prints the inertia with respect to the center of gravity. Both tables are now labelled accordingly, and a note is added to mention that inertia is not printed in tensorial form.

Threshold for thin-shell thickness switched to 0.01
The threshold for printing WARNING # 1265 identifying thin-thickness shells is now changed from 0.001 to 0.01.

Auto contact for implicit nonlinear analysis (beta)
Auto-contact is now supported for implicit nonlinear analysis. Similar to auto-contact in explicit analysis, auto-contact for implicit analysis is activated by setting the *TYPE* field to AUTO. The *ACTIVA* continuation line is used to activate auto-contact for certain surfaces, and *DEACTIVA* continuation line is used to deactivate certain contact surfaces from auto-contact. The *PCONT* continuation line can be used to correspondingly activate contact properties for contact interfaces.

Nonlinear damping now supported in nonlinear static and nonlinear transient analysis for `JOINTG`
Nonlinear damping can now be defined via *PROPERTY*=NDAMP on the `PJOINTG` Bulk Data Entry. A force-velocity curve can now be applied to each degree of freedom associated with the joint to define nonlinear viscous damping.

Creep modeling now available for gasket elements
Creep behavior can now be modeled for GASKET elements using `MGASK` and `MATVP` Bulk Data Entries.

Modeling a curved beam or pipe element now supported

A curved beam element can now be modeled using the CBEND element and PBEND property Bulk Data Entries. CBEND and PBEND can be used for the following types of linear analyses: linear static analysis, normal modes analysis, frequency response analysis (direct and modal), random response analysis, and linear transient analysis (direct and modal).

Hydrostatic fluid element modeling available via MONVOL

MONVOL is now supported for implicit nonlinear large displacement analysis. MONVOL can now be used for specific use cases such as airspring modeling and modeling hydrostatic fluid elements. It is supported for 3D, axisymmetric, and plane-strain elements. The element set can be used to define a cavity fully filled with ideal gas/hydraulic fluid. Both hydraulic and pneumatic fluids can be modeled, and the cavity volume, applied pressure cavity pressure, and energy values can be output.

Mixture of axisymmetric and plane-stress elements in the same contact interface

A CONTACT/TIE interface can now contain axisymmetric elements (CQAXI, CTAXI) on one side and plane stress elements (CQPSTS, CTPSTS).

Explicit Dynamic Analysis

Implicit explicit chaining analysis (beta)

Explicit subcases can now continue from a preceding implicit subcase using the CNTNLSUB entry. This makes various applications available, for example, models where bolt pretensioning can be conducted first in the implicit subcase and subsequently the loading can be applied in the explicit subcase.

Low-density foam material modeling now supported

Low-density foam material can now be modeled by specifying LDFOAM material model in the MATHE Bulk Data Entry. The low-density foam material model is intended for highly compressible low-density foams with significant rate sensitive behavior. It requires the direct specification of uniaxial stress-strain curves at different strain rates for compressions. Optionally, uniaxial stress-strain curves at different strain rates for tensions can also be defined.

Out of plane Poisson's ratio now available in MAT8

Out of plane Poisson's ratio can now be defined using the *NU13* and *NU23* entries on the MAT8 Bulk Data Entry. Defining these parameters allows for precise internal force integration, better description of necking and better consistency with 3D orthotropic element results.

GSETID on TIE supported

The *GSETID* field on the TIE entry is now supported for explicit analysis. It is used to identify a subset of the secondary surface for which corresponding TIE contact is created regardless of the *SRCHDIS* value.

5-noded CPYRA element supported

The 5-noded CYPRA element is now supported for explicit analysis. It contains 5 integration points without hourglass control.

MPC support

Multi-point Constraints (MPCs) are now supported for explicit analysis. Multiple MPCs with the same ID are currently not supported for explicit analysis.

Shell offset support for contact in explicit analysis
Shell offset is now supported for contact interfaces in explicit analysis.

Corner and gauss stress/strain support
The following are now supported for explicit analysis:

- Solid elements: Support both corner and gauss stress and strain results in the H3D file.
- Shell elements: Support gauss stress and strain results in the H3D file.

Plasticity and damage behaviour of `JOINTG` elements
Plasticity and damage behaviour of `JOINTG` elements is now supported for explicit analysis. Four result items unique to plasticity and damage related analysis are available: JOINTG Plastic Displacement, Potential, Damage Index and Mode-mix ψ. These results are only available for explicit analysis in H3D format and supported for CARTROTA and CARDAN elements.

Rayleigh damping support for `JOINTG` SLIPRING
Rayleigh damping is now supported via `PARAM,ALPHA1` and `PARAM,ALPHA2` for `JOINTG` SLIPRING.

TSTIME field on `LOADJG` and `MOTNJG` now supported
The *TSTIME* field on `LOADJG` and `MOTNJG` to switch time duration for loading/motion curves between Total time (TOT) and Subcase time (SUB) is now supported for explicit analysis.

`TTERM` can now be based on subcase time or total time in explicit analysis
`TTERM` in explicit analysis can now be based on either a subcase time using `TTERM(SUB)` or for the total simulation using `TTERM(TOT)`.

Hyperelastic material failure now supported in explicit analysis
Failure for hyperelastic materials for both solid and shell elements is now supported in explicit analysis. In addition, the new generic failure model can be used to introduce element deletion and define multiple failure criteria via *GENE1* criterion in `MATF` Bulk Data Entries. The *GENE1* criterion includes the following failure allowables: minimum hydrostatic pressure, maximum hydrostatic pressure, maximum principal stress, maximum von Mises stress, Tuler-Butcher criterion, maximum principal strain, maximum von Mises equivalent strain, maximum volumetric strain, and maximum shear strain.

`GPSTRESS` and `GPSTRAIN` results for explicit analysis now supported
`GPSTRESS` and `GPSTRAIN` results are now supported for explicit analysis (NLEXPL) in Hyper3D format. Corner stress along with `GPSTRESS` and `GPSTRAIN` are also now supported for second order elements like TETRA10 for explicit analysis.

Section force and moment results now supported in `THIST`
Time history output request (`THIST`) now supports force and moment results on selected sections. The desired outputs can be requested via the *SECTION* continuation line on the `THIST` Bulk Data Entry. Supported only in explicit dynamic analysis.

Main surface definition on ACTIVA/DEACTIVA when secondary surface is set to ALL
The main surface definition can now be specified on *ACTIVA*/*DEACTIVA* when secondary surface is set to ALL. With this option you can create or remove auto-contact between a single surface and the rest of the model.

Exponential decay for friction now supported
The *FRICTION* continuation line on `PCONT` Bulk Data Entry is now available to define exponential decay for friction. There are three options to define exponential decay:

- Keyword EXPDCAY is specified on field 3 of the *FRICTION* continuation line, and subsequent fields define the sliding friction (*MUs*), kinetic friction (*MUk*), decay factor (*Dc*), and relative sliding velocity (*Vrel*).
- Keyword EXPDCAY is specified on field 3 of the *FRICTION* continuation line, and keyword TESTDAT is specified on field 4, and test data is specified as follows in the subsequent fields: *MU1*, *MU2*, *VREL2*, and *MU∞*. This test data is used to fit the value of the decay factor *Dc*.
- Keyword DEPEND is specified on field 3 of the *FRICTION* continuation line, and a 2-dimensional table is specified below if friction is only dependent on relative velocity (*Vrel*), and a higher dimensional table is specified if there are more dependencies, such as contact pressure. This higher dimensional table is internally converted to a `TABLEMD` entry.

Noise and Vibration

Enhanced rib-detection to automatically handle element normals for panel-based ERP output
Advanced algorithms were already available to automatically flip element normals when required for panel-based ERP output (for instance, when there are panels with intersecting ribs). The algorithm is now enhanced to handle a wider variety of situations, including when an edge is connected to more than two faces.

Multiple `PEAKOUT`s for same grid and DOF
Multiple `PEAKOUT` Bulk Data Entries with the same ID are now supported for the same grid and DOF. The results are added up during the peak identification process.

`JOINTG` contribution in `GPFORCE` for frequency response
The contribution from `JOINTG` is included in `GPFORCE` output in the `.gpf` file for frequency response analysis.

Viscous to structural damping conversion now supported
`PARAM, VISC2MAT,`<value> can now be used to convert viscous damping matrix to material damping matrix. The specified value is also used to scale the final material damping matrix. Supported only for complex eigenvalue analysis.

`PARAM` to switch `MATPE` ON or OFF now available
`PARAM, MATPE` is now available which can be used to switch between fluid element flags of `PSOLID`. `PARAM, MATPE, OFF` switches `PSOLID` fluid element flag from PORO to SMECH and vice-versa for `PARAM, MATPE, ON`.

Peak rate output support for random response resultant
Peak rate is now output as a part of the `RESULTANT` output for SECTION-based resultants of random response analysis. This is now available in addition to the previously available zero crossing rate.

`PEAKOUT` now supported for linear transient and steady state analyses
`PEAKOUT` is supported for transient and steady state analysis. `PEAKOUT` peak finding is supported for displacement, velocity, acceleration, pressure, and `ERP`. Peak results output is tested for displacement, velocity, acceleration, pressure, stress, strain, force, `GPFORCE`, `SPCF`, applied force, `ESE`, and `ERP`. It is supported for H3D, OP2, and PUNCH.

Global-local analysis supported for random response analysis
Global-local analysis (sub-modeling) is now supported for random response analysis. This is supported by mapping the base frequency response displacement, velocity, and acceleration

results from the global model H3D file to the local sub-model. The global-local interface is specified via the SPCD entry with M option selected for mapping; the global model H3D file is specified using ASSIGN, H3DRES. A random response analysis subcase can be defined in the local model so that the frequency response results calculated in the local model based on the mapped global model results are used for subsequent random response calculations.

Multiple RADSND now supported using RADADD
Multiple RADSND Bulk Data Entries with different IDs can now be referenced by a RADADD Bulk Data Entry and this can be referenced by a RADSND command in a subcase. This is also supported for Optimization, wherein the individual RADSND IDs from RADSNDs included on a single RADADD Bulk Data Entry can be referenced in responses.

Sound Pressure Level (SPL) Optimization
Topology optimization is now supported for Radiated Sound Analysis (RADSND). The exterior sound pressure response can now be defined using *RTYPE*=FRPRES on the DRESP1 entry along with specifying the RADSND ID to which the response applies on the third field of the *EXTN* continuation line.

MultiPhysics

Applied pressure output from OptiStruct-AcuSolve co-simulation
Applied pressure in OptiStruct during the OptiStruct-AcuSolve co-simulation is now available using the OLOAD request.

Axisymmetric elements are supported in electrical analysis
Axisymmetric elements are supported in electro-thermal and electro-thermal-structure coupling analyses. Currently first order elements CQAXI and CTAXI elements are supported. Contact with axisymmetric elements is also supported.

Electrostatic subcase now available
A new standalone subcase for electrostatic analysis is now supported and can be defined using ANALYSIS ESTAT. ESTAT can be used for finding electric potential distribution in a dielectric material and evaluating electrostatic force on the structure. Electric potential distribution and electrostatic force can in turn be used as external loading in structural analysis.
Various types of inputs can be defined in the new subcase such as potential, nodal charge, area charge density and volume charge density. Relative permittivity, absolute permittivity and vacuum permittivity can also be defined via MAT1PT, MAT2PT and PARAM, VAPMTV respectively. VOLTAGE, ELECFLUX, ELECFIELD, GPCHARGE, OLOAD, SPCCHARGE, MPCFORCE (output request for MPC charge) and ESTATFORCE are some of the output requests available for the new subcase.

Electrostatic force output for electrostatic analysis is now supported
Electrostatic grid and resultant forces can now be requested for electrostatic analysis loadcases. These results are supported in H3D and OPTI format. In the case of an OPTI result request, results are output in ASCII formats including OptiStruct results format (.estatf and .estatresf) and CSV format (_estatf.csv and _estatresf.csv).

Supersonic aeroelasticity
Aeroelasticity is now supported for the supersonic regime using the Constant Pressure Method (CPM). This is now supported for trim, flutter, and divergence analyses.

Optimization

LGELIM supported for optimization

The constraint elimination method (*RIGID*=LGELIM) is now supported generally for optimization. If rigid element grids are in the design space for shape optimization, LGELIM is switched automatically to LAGRAN.

Level-set method supported with DDM

Domain Decomposition Mode (DDM) is now supported for level-set method.

New type of geometric responses now supported in structural optimization

Geometric response and geometric deformation response are now supported in structural optimization. These responses can be defined using `DRESP1` Bulk Data Entries. Geometric response is a load-independent response that can be selected using *RTYPE*=GEORESP. These responses are functions of the grid locations specified in the *ATTi* fields of the `DRESP1` entry. There are 19 types of geometric responses available.

Geometric deformation responses can be obtained from a linear static analysis. These responses are functions of the deformed grid locations and are selected using *RTYPE*=GEODEFR. The grids of interest are specified in the *ATTi* fields of the `DRESP1` entry. There are 19 types of geometric deformation responses now available.

Multiple user-defined milling constraints now supported in topology optimization

Multiple user-defined milling constraints can now be defined through `MILLU`. Milling constraints can be defined in 3 ways using access angle, defining the dimensions of milling bit and head, and defining access angle using access angle and radius of the mill bit. Multiple access directions can now be defined using the `MILLU` Bulk Data Entry.

Shape fraction response now supported in vertex-based free-shape optimization

Shape fraction response is now supported in vertex-based free-shape optimization. It is activated using the *RTYPE*=SHAPFRAC option on the `DRESP1` entry. The *ATTI* field can be used to specify `DSHAPE` IDs or it can be left blank in which case all `DSHAPE` entries are included. The *ATTA* field can be set to TOTAL(default), MAGNI, NORM, or blank. For the TOTAL option, the shape changes in the X, Y, and Z directions are added together. The MAGNI option uses the resultant magnitude of the shape change, and the NORM option only considers the shape change in the direction normal to the surface.

New adaptive time selection method now available in ESL

A new adaptive time step selection method is now supported in ESL and can be applied via *TYPE* = 3 in the `ESLTIME` Bulk Data Entry. The time steps are selected such that the target response curve is fitted by the piecewise linear curve spanned by the ESL-times, thereby enhancing the performance of the ESL method.

General

`POWERFLOW` through `SECTION` for transient analysis

`POWERFLOW` output is now available through `SECTION`s defined in transient analysis. This is currently only supported for H3D output.

GPU support for FASTFR

GPU is now supported for *FASTFR*. Any solution for which the frequency-response part is performed using *FASTFR* can now be parallelized with GPUs.

On-the-fly H3D output enhancements

The following results are now added in on-the-fly H3D output:

- CROD elements: Stress, Strain, Mechanical Strain, Thermal Strain, Plastic Strain, and Equivalent Plastic Strain
- CBEAM elements: Stress and Strain
- Shell layer results are now supported for on-the-fly H3D output for implicit nonlinear analysis

ESE support for JOINTG

Element Strain Energy (ESE) results are now supported for JOINTG elements in the regular and on-the-fly H3D files.

Friction for CARTESIA joint on JOINTG

The *FRICTION* continuation line can now be used to define friction for JOINTG type CARTESIA. It can either be defined in the tangential direction (using a single DOF) or in a tangential plane (using 2 DOFs). The tangential direction degree(s) of freedom can be defined using the *TDOF* field and the normal direction degree of freedom can be specified using the field *NDOF* in the *FRICTION* continuation line on the PJOINTG Bulk Data Entry. Frictional coefficient can be defined using the *MU* field. The tangential direction can be defined for a single component or spatially or there can be two separate directions, one for each tangent direction.

SPC force output for cyclic symmetry analysis

SPC force output is supported in cyclic symmetry for linear static and normal modes analysis in the regular H3D file. It is also supported in cyclic symmetry for implicit nonlinear analysis in the regular H3D file and the on-the-fly H3D file.

NLPCI support for NLDEBUG,NLP2NLC

NLPCI is now supported for NLDEBUG,NLP2NLC. When the model has an NLPCI sharing an ID with an NLPARM, the NLPCI entry is assigned the same ID as the internally generated NLCTRL.

ASSIGN,HFILE can refer to an H3D file for transient temperature input

In addition to the previously supported PUNCH file, the ASSIGN,HFILE entry can now refer to an H3D file to identify transient temperature input.

SORT2 support for FORCE output in OPTI format

SORT2 is now supported for FORCE output in OPTI format for linear static and nonlinear static analysis (SMDISP and LGDISP).

Random response output enhancements for HDF5

The following random response output enhancements are now available in the HDF5 format:

- 1D element FORCE
- DISP, VELO, ACCE
- STRESS, STRAIN
- Composite Ply STRESS, STRAIN
- SPCFORCE

REPCASE support for linear solutions

REPCASE is now supported for linear analysis, such as normal modes analysis, complex eigenvalue analysis, frequency response analysis, transient response analysis, and random response analysis.

Create SET by INCLUDE functionality

The functionality to create a SET of elements for each INCLUDE file specified in the model is now supported via PARAM, SETINC,YES. These SETs are output to the H3D file. This is supported for all types of elements including rigid elements.

Results output on only the SKIN of the component

The MODEL card is now enhanced with the SKIN option to generate results only on the surface skin of the model.

DISPLACEMENT output now supported on skin of component

Enhanced DISP output via the SET Bulk Data Entry now supports SKIN as a new option to output displacements only on the exterior skin of the component.

Control number of digits printed in OPTI format

The number of digits of the results printed to the OPTI file can now be controlled via PARAM, DIGIT. The maximum limit is now 12 digits for any result requested.

Math equation support for symbolic substitution

Math equations are now supported for symbolic substitution. The equations can be defined anywhere in the input deck, and both %setrepsym and %defrepsym are supported. The equation syntax is consistent with how equations are supported for DEQATN; all the operators supported for DEQATN are also supported for symbolic substitution. However, unlike DEQATN, the arguments do not have to be explicitly provided. Equations can use other equations as arguments. The same rules of general symbolic substitution also apply to equations, such as the order/sequence of the cards and variables defined in the input deck matter, and the variables can be redefined in the model and are always evaluated based on the current value of the variables. Below is an example:

```
%defrepsym fac=2.0
%defrepsym dens=0.007*fac
%defrepsym mass=0.2*dens

MAT1           1210000.0        0.3     %dens%
CONM2         51       32       0%mass%            0.0     0.0     0.0
```

Symbolic substitution now supports TTERM

TTERM can now be defined using symbolic substitution. %defrepsym and/or %setrepsym can be used with TTERM in the control section of the input deck.

Enhanced support for PARAM, POST,-5 - grid renumbering and optional job termination

Original grid numbering can now be retained in the .k.op2 and .m.op2 files to maintain consistency between these OP2 and FEM files; this is now supported via the *RENUM* option in PARAM, POST when the value -5 is used. Additionally, the job can now be aborted after the .k.op2 and .m.op2 files are written by using the *STOP* option.

Enforced loading via RSP file now supported in transient analysis

Enforced loading via external RSP files are now supported in transient analysis. The enforced loads can be enforced displacements, velocities, and accelerations. The external load can now be specified on the TLOAD1 card by setting the *TID* field to EXTLOD and the corresponding *TYPE* field can be set to DISP, VELO, and ACCE. The ASSIGN, EXTLOD option can then be used to identify the external RSP file and the corresponding mapping CSV file and the ID of this ASSIGN, EXTLOD entry can be referenced on the *EXCITEID* field of the TLOAD1 entry.

Unit system specification of external data now supported in INISTRS/INIPS

The unit system of the external data can now be specified using the *UNITS* continuation line in INISTRS or INIPS Bulk Data Entries. In case the length unit specified in the INISTRS/INIPS Bulk Data Entries is different from the current model's length unit, a model scaling is conducted for the

external H3D model to match the length unit to the current model. In the case of model scaling, the new *RELOC* continuation line is mandatory along with the *UNITS* continuation line.

Shell layer results now available in on-the-fly H3D file
Shell layer results are now supported in on-the-fly H3D output format for implicit nonlinear analysis.

SET for TIME now supports specifying time interval and range
The `SET` entry, which defines the set of time points at which the incremental nonlinear output is requested, now includes *THRU* and *BY* functionalities. The time interval can be specified using *BY* and the range of points can be specified using *THRU* in the `SET` Bulk Data Entry.

Sortable tables in OptiStruct online help
New table sorting feature for large tables.
This feature has been implemented in:

- Run Options
- Entries and Format
- Solvers versus Supported Solution Sequences

Note:

Platform MPI support discontinued
As of OptiStruct 2025, the support for Platform MPI has been discontinued. This also implies that the dedicated OptiStruct Platform MPI executable will not be packaged in the HWSolvers installation anymore, and the "`-mpi pl`" run option will not be available anymore. Please use the available Intel MPI (Windows and Linux) and Open MPI (Linux only) options for MPI jobs.

Resolved Issues

- When AMSES or Lanczos is used for fluid-structure PFPATH analysis, the fluid modal space calculation is no longer conducted twice.
- The performance of steady-state analysis is enhanced in SMP parallelization mode.
- A crash no longer occurs due to lack of MUMPS memory in a certain model with electro-thermal analysis.
- There is no longer an issue with the joule heating calculation for 2nd order tetra and penta elements.
- There is no longer an issue with stress results when `PARAM,CURVSHL2,THICK` is used in models with in-plane bending.
- A programming error no longer shows up when the frequency range is specified on the `EIGRL` entry with `PARAM,AEMESH,YES`.
- A programming error no longer shows up when MPC is present in a nonlinear cyclic symmetry model.
- A model with MCIRON no longer fails to converge when stress is less than yield limit.
- Linear heat transfer results, such as grid temperatures, are now correct when run in DDM mode.

- ESE output is now active for JOINTG in on-the-fly H3D file. Additionally, ESE output is now supported for JOINTG.
- GPFORCE output performance to the H3D file for normal modes analysis is now efficient when compared to similar output for linear static analysis for the same model. Significant speedup is now available for GPFORCE output for normal modes analysis.
- The correct density field is now initialized for Optimization restart with milling constraint.
- Certain Intel MKL libraries are no longer missing for Compose, and OptiStruct runs which require Compose no longer fail.
- A powerflow optimization model containing both SECTION entries for powerflow and ERPPNL cards for ERP no longer fails.
- Radiated sound pressure calculated based on PLOTEL from superelement is now accurate.
- A model with Surface-to-Surface (S2S) CONSLI contact with *DT*=1.0 no longer has an issue detecting contact.
- A model showing inefficient performance in CONTRES module and worse performance in DDM when compared to SMP is now fixed.
- Correct MFLUID results are now generated when increasing the frequency range on EIGRL for Lanczos eigen-extraction.
- A programming error no longer shows up if the PCONT entry is not referenced on auto-contact in an explicit model.

Altair Radioss 2025 Release Notes

Highlights

- Continuous development for the LS-DYNA format reader.
- First option /FUNCT_PYTHON where Python script can define function values.
- Option to control solid element distortion ($I_{control}$=1) to improve solid element stability and avoid negative volumes.
- Improvement of rigid body (/RBODY) stability. This modification may lead to some small differences from previous versions.
- New table sorting feature for large tables.

New Features

LS-DYNA Format Reader

Additional LS-DYNA format reading capabilities

- *CONTROL_HOURGLASS
- *INITIAL_STRESS_SOLID
- *MAT_ADD_THERMAL_EXPANSION
- *MAT_005 (SOIL_AND_FOAM)
- *MAT_073 (LOW_DENSITY_VISCOUS_FOAM)
- *MAT_111 (JOHNSON_HOLMQUIST_CONCRETE)

- *MAT_169 (ARUP_ADHESIVE)
- *MAT_183 (MAT_SIMPLIFIED_RUBBER_WITH_DAMAGE)

Loadcase and Contact Interface

- /DFS/DETPOINT/SET: option to define detonation points applied on a set of nodes (/SET or /GRNOD).
- /FUNCT_PYTHON: new possibility to get the current time and time step in a Python script.
- State output (/STATE/SPHCEL/FULL) and input (/INISPHCEL/FULL) to initialize Smooth Particle Hydrodynamics.

Material and Failure Models

- /MAT/LAW169 (ARUP_ADHESIVE): new material law for adhesive material law to simulate adhesive with ability to take into account both cohesive and damage behavior, tailored for materials that exhibit a mix of elastic, plastic, and brittle failure characteristics (for solids).

Enhancements

LS-DYNA Format Reader

- *DATABASE_HISTORY_{OPTION}_SET: keyword reading improvement for all input types.
- *DATABASE_HISTORY_BEAM: specific beam element type was missing in the time history file.
- *DEFINE_COORDINATE_NODES: reading improvement and corrections
- *MAT_002 (ORTHOTROPIC_ELASTIC): this material is now mapped to /MATLAW93 for better accuracy.
- *MAT_009 (NULL): null material *MAT_NULL with *EOS was not correctly read.
- *MAT_077_O (OGDEN_RUBBER): the number of material parameter pairs is extended from 5 to 8.
- *MAT_181 (SIMPLIFIED_RUBBER/FOAM): improvement of the material reading and mapping.
- *SECTION_SHELL: reading improvement in case *NIP* > 10.
- *SET_PART_ADD: correction of set reading in case a set type has the same identifier.
- *SET_PART_LIST: nodes from SPH elements were not included in the set.

Loadcases and Contact Interfaces

- /CLOAD, /PLOAD: the concentrated load and pressure load can be a function of node displacement or velocity.
- /INIVEL/AXIS, /INIVEL, /INIVEL/FVM: new option to define the starting time; from defined value or sensor (/SENSOR).
- /INIVOL:
  - New option *ICUMU* = -1 defines subtractive input of volume fraction. The current surface is used to fill the element and if sum of ratio is greater than 1, then it is subtracted from the previous initial volume setting.
  - After all execution of initial volume options, the unoccupied sub-volume is now automatically filled with the prevalent phase (*VFRAC* > 0.9). It was filled with phase 1 in the previous version.
  - Compatible with 2D solid elements for axisymmetric and plane strain analysis.

- /INTER/TYPE1:
  - Error message added in the Starter in case of incompatible boundary condition in the Starter.
  - Additional check in the Starter for the tied FSI contact to verify if the main side is defined in Lagrangian domain and the secondary side in the ALE domain.
  - Global improvement in the search algorithm of the fluid versus structure tied contact
- /INTER/TYPE2:
  - Modification of the output to consider the loading direction and display positive value in tensile and negative value in compression.
  - Some corrections have also been made for the contact force outputs. The results will be different from the previous version.
- /INTER/TYPE24, /INTER/TYPE25:
  - New warning message in case *INACTI* flag is set to a non-expected value. The flag *INACTI* is reset to default value.
  - Correction of Starter segmentation violation when /INTER/SUB is used.
- /INTER/TYPE7, /INTER/TYPE11:
  - Improvement of the behavior of *Irem_gap*=2 with I_{gap}=3. Some contacts were not correctly removed.

Material and Failure Models

- /FAIL: failure model ID of the deleted/ruptured element is now printed in the Engine output file.
- /FAIL/FLD: new failure model based on Forming Limit Diagram which handles nonlinear strain paths. This failure model is used in sheet metal forming.
- /FAIL/GENE1: improvement of the compatibility between parameter *dtmin* and constant nodal time step (/DT/NODA/CST).
- /FAIL/JOHNSON: new failure options I_{fail_so}=3 and I_{fail_so}=4 for solid element dedicated for ballistic or high-speed impact applications.
- /MAT/LAW1, /MAT/LAW42, /MAT/LAW62, /MAT/LAW69: increased contact stiffness in case the element distortion control is activated.
- /MAT/LAW2 is now compatible with all Equation of State (/EOS).
- /MAT/LAW36 is now compatible with Equation of State (/EOS).
- /MAT/LAW42: new capability to define up to 10 shear hyper elastic moduli and material exponents.
- /VISC: viscosity models can now be used for any material laws compatible with shell elements.

Element and Properties

- /PROP/SOLID, /PROP/SO_ORTH, /DEF_SOLID: new flag $I_{control}$=1 to activate the solid element distortion control and internal contact in the solid element to avoid negative volume.

Animations and Time History Output

- `/TH/SURF`: new output for the pressure, mass flow and fluid velocity through a surface coming from Element Boundary Condition `/EBCS/INLET`, `/EBCS/FLUXOUT`, `/EBCS/NRF` and internal surface of ALE/EULER domain defined with `/SURF/SEG`.

Starter and Engine

- Update and improvement of the stacksize setting.
- New option `-python` to explicitly authorize usage of Python script in Radioss.

Verification Models

- New verification problem for failure model `/FAIL/BIQUAD` (RD-V: 0710)
- Update to the fabric material verification problem (RD-V: 0230)

Resolved Issues

LS-DYNA Input Format

- `*CONSTRAINED_INTERPOLATION`: element stability issue with large time step.
- `*CONTACT`: mapping for the parameters SST and MST (optional contact thickness).
- `*CONTACT_AUTOMATIC`: reading correction if *SOFT*=2.
- `*DEFINE_COORDINATE_NODES`: reading correction if *dir*=Y. The coordinate system was not well oriented.
- `*DEFINE_TRANSFORMATION`: reading improvement if the transformation type is not recognized or valid.
- `*ELEMENT_DISCRETE`: reader improvement if there is no local coordinate system (`*DEFINE_SD_ORIENTATION`) defined in the discrete element.
- `*INITIAL_VELOCITY`: node set that are exempted from the initial velocities (*Exset_ID*) is now correctly read.
- `*MAT_015 (JOHNSON_COOK)`: reading correction if *E*=0.
- **SET*: reading correction for the empty set (**SET*).

Loadcases and Contact Interfaces

- `/ALE/GRID/FLOW-TRACKING`: behavior improvement in case of rotational transformation (`/TRANSFORM/ROT`).
- `/ALE/MAT`, `/EULER/MAT`: correction of communication issue for ALE or Euler material formulation with SPMD; HMPP version and for specific model.
- `/EBCS/INLET`: compatible with `//SUBMODEL`
- `/INIVOL`: incorrect initialization with specific model with `/MAT/LAW151` and linear tetrahedron (`/TETRA4`)
- `/INTER/TYPE1`: ALE tied interface is now compatible with 2D analysis.
- `/INTER/TYPE21`: correction of contact interface checking, incorrect minimum contact gap value was printed in the Starter output file.
- `/INTER/TYPE24`, `/INTER/TYPE25`: memory allocation issue which leads to a segmentation fault in the Engine restart when the output option `/H3D/NODA/CSE_FRIC` was used.

- /INTER: improvements and corrections of the contact interface surface update in case of element failure.
- /RBODY: stability improvement of the rigid body with 2 secondary nodes in case of high rotational velocity.
- /RETRACTOR/SPRING:
  - Improvement of the seatbelt behavior if only fct_ID_1 is defined with a null or negative slope.
  - New error message if only fct_ID_1 and fct_D_2 are defined and have a null or negative slope.
- /RWALL: memory allocation issue.
- /SECT/PARAL:
  - Reading for /SECT/PARAL with specific model.
  - Correction of warning message for the section forces when the preload option (/PRELOAD) is used in the model.
- /SET/GENERAL:
  - Surface build corrections and improvement.
  - Correction for the 2D surface defined with segments.
  - Reading improvement for the SPH elements.
  - One attribute was missing for the set of nodes with the key *NODENS*.
  - Issue with surface building for 3 node shell elements (/SH3N) with adaptive meshing (/ADMESH).
  - The nodes from SPH defined by parts were not correctly extracted from the general set.
  - Reading correction for the set used in the drawbeads contact /INTER/TYPE8.
- /SKEW, /FRAME: reading correction in case the reference direction is written with a lower-case letter.
- /SLIPRING/SHELL: correction of animation file output for specific model computed with SPMD/HMPP version.

Material and Failure Models

- /FAIL/FRACTAL_DAMAGE: initialization may be wrong with SPMD/HMPP version in win64 platform.
- /MAT/LAW2: improvement of the thermal behavior for the elastic-plastic material laws.
- /MAT/LAW25: correction of wrong behavior for tensile plastic damage in direction 1 with solid elements
- /MAT/LAW83: strain rate was not correctly computed for F_{smooth}=1 and specific model. Strain rate value was also not output in solid element time history.
- /MATLAW58, /MAT/LAW19: reading improvement for the single precision version.
- /MAT/LAW62: stability improvement for specific loadcase.
- /MAT/LAW76: correction of Radioss Engine for specific model.
- /NONLOCAL/MAT:
  - Improvement of the post-failure behavior of non-local elements.

- Correction of parallel arithmetic behavior with non-local failure formulation.

Element and Properties

- `/PROP/TYPE29 (USER)`: memory allocation correction for the UVAR buffer for the user property interface.
- `/PROP/TYPE3 (BEAM)`:
  - Correction of memory allocation issue if there is a failure model defined for the material used in the beam.
  - Starter no longer fails when material with failure model (`/FAIL`) is defined on the material set on a beam.

  .
- `/PROP/TYPE45 (KJOINT2)`:
  - The joint main direction is defined from node N1 and N2 in case *Skew_ID*$_1$ and *Skew_ID*$_2$ are defined.
  - This modification may lead to some results differences compared to previous versions from specific models.
- `/PROP/TYPE1 (SHELL)`, `/PROP/TYPE9 (SH_ORTH)`, `/PROP/TYPE10 (SH_COMP)`: flag I_{pos} is added in all shell properties to model shell offset from the mid plane.

Animations and Time History Output

- `/ANIM/BRICK/TENS`, `/H3D/SOLID/TENS`: all integration points are now output in the animation file (`/ANIM`) and the native `.h3d` file (`/HD3`) for the linear tetrahedron with I_{tetra4}=1.
- `/H3D/BEAM/TENS/STRESS`, `/H3D/BEAM/TENS/STRAIN`: memory allocation correction.
- `/H3D/QUAD`: correction of vector output for the 2D solid element.
- `/H3D/SHELL` corrections:
  - "Erosion status (s)" output.
  - `/H3D/SHELL/DAMA` output for `/MAT/LAW25`.
  - `/H3D/SHELL/TENS/STRESS` output. Some unexpected PLY was output.
  - `/H3D/SHELL/FAIL` output.
  - `/H3D/SHELL/FAILURE/*` for specific case.
- `/H3D/SPRING/FORC`, `/H3D/BEAM/FORC`, `/H3D/TRUSS/FORC`: the 1D element force and moment writing in the `.h3d` file is now readable.
- `/H3D/NODA/GPS/TMAX`, `/H3D/NODA/GPSTRAIN/TMAX`: maximum stress values could decrease according to time for specific models.
- `/TFILE`, `/ABF`: time history data for the parts were missing in case a non-existing part is set in the `/TH/PART` card.

Starter and Engine

- Radioss Starter no longer fails with the option `-output=[path]` if the length of Starter output file name with the full path is longer than 100 characters.
- `/DT/NODA/CST`: parameter *initial_mass_ratio* is now correctly considered for multiple runs.

Altair MotionSolve 2025 Release Notes

Altair Multibody Solutions 2025 Release Notes

Highlights

The 2025 release of Altair Multibody Solutions introduces several enhancements and bug fixes to MotionView, MotionSolve, and their extensions. Here are some of the key highlights:

- Save system definition to the Entity Browser (MotionView)
- Deformable curve on a Polybeam (MotionView)
- Improvements to Frequency Response Analysis (MotionSolve)
- CD Tire updates (MotionSolve/Vehicle Tools Extension)

New Features

Save System Definition to the Entity Browser (MotionView)

MotionView's Entity Browser contains a list of entities that can be added to the model. It is now possible to save systems to the Entity Browser for quick retrieval and reuse.

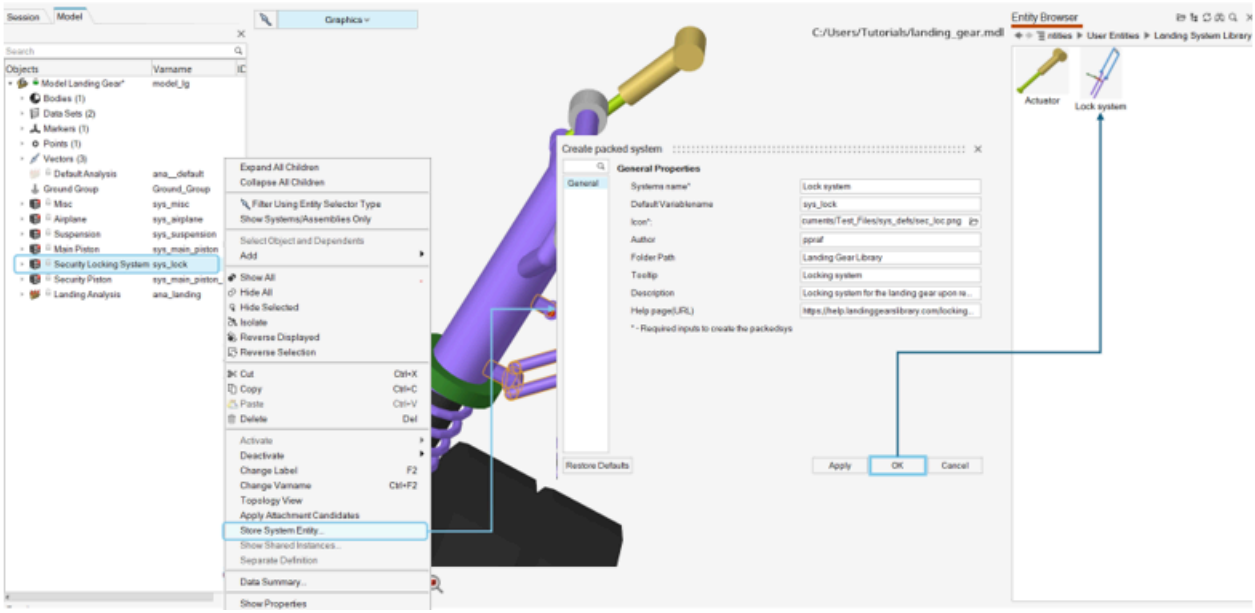

Figure 73:

Select a system in the Model Browser, right-click, and select **Store System Entity...**. In the subsequent dialog, fill in system details, an icon image, and click **OK**.

- The system is listed under the User Entities folder.
- Provide a folder path to organize the stored systems.
- Double click on the stored system to insert it into the active model.

Deformable Curve on Polybeam (MotionView)

In the previous release, MotionView and MotionSolve added support for the deformable curve contact, which can be used to model pipes, cables, and tubes in contact. In this release, MotionView further enhances this functionality by providing an option to add a deformable curve with graphics when creating a polybeam entity. This improvement streamlines the setup of models with slender elements, enabling faster configuration.

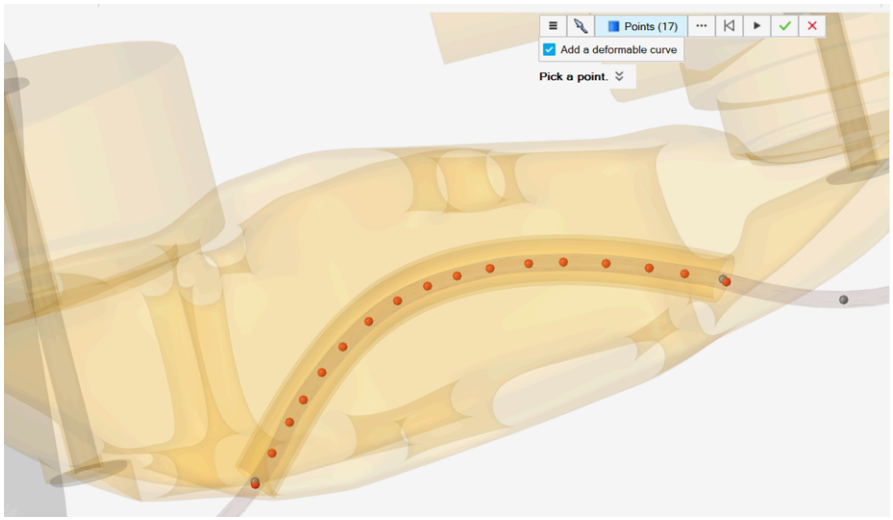

Figure 74:

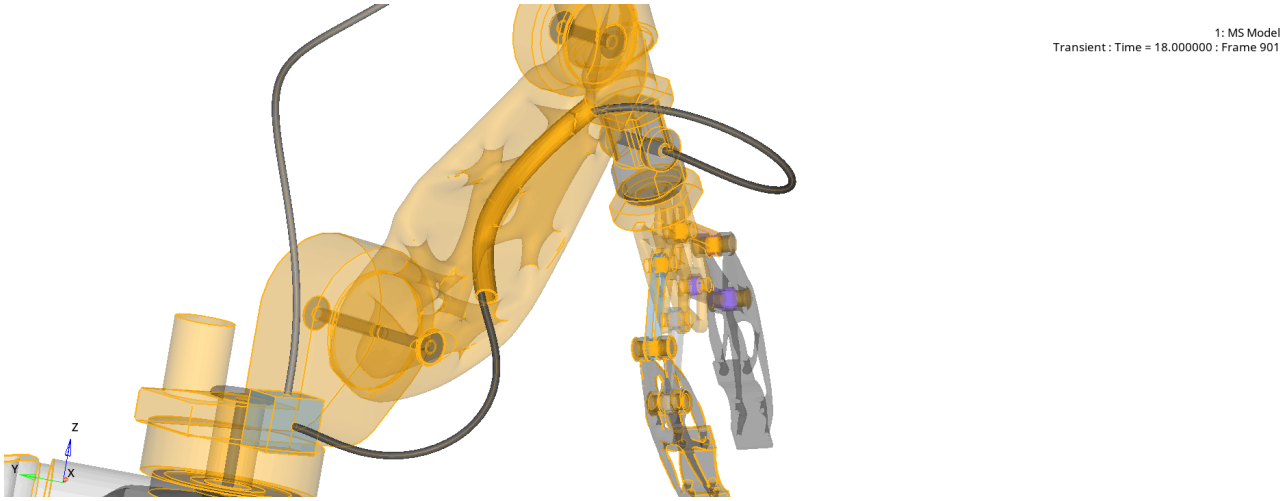

Figure 75:

Enhancements

Import Markers and Geometrical Curves from Catia (MotionView)

The CAD import functionality has been enhanced to read any existing marker and geometrical curve from Catia V5.

Improvements to Frequency Response Analysis (MotionSolve)

In this release, the Frequency Response Analysis in MotionSolve introduces additional support for flexible bodies:

- Frequency-dependent modal participation factors.
- Frequency-dependent kinetic energy, dissipative energy, and strain energy distribution.
- Frequency-dependent transfer path analysis.

Write mkb Matrix in Linear Analysis (MotionSolve)
In this release, MotionSolve includes support for exporting of the mass, stiffness, and damping matrices of a linearized system.

Fraunhofer CD-Tire: Resizing (Vehicle Tools Extension)
The Vehicle Tools Extension now supports the Resizing feature for CDTire from Fraunhofer ITWM. The resizing feature allows a tire model to be automatically resized based on reference and target tire and rim specifications.

Fraunhofer CD-Tire: Support for MF++ (Vehicle Tools Extension)
The Vehicle Tools Extension now supports the Fraunhofer MF++ tires. CDTire/MF++ is a temperature enhanced Magic Formula for coupling to CDTire/Thermal in advanced handling applications.

Fraunhofer CD-Tire: Licensing Changes (MotionSolve)
MotionSolve has updated the license check-out logic for CDTire and the CDTire resizing feature. MotionSolve supports the usage of the CD-Tire resizing feature. To use this feature, your Altair license must include both the CDTire_MSolve (regular CD Tire) and CDTire_MSolve_RS (CD Tire with resizing) features. MotionSolve first checks out the CDTire_MSolve feature, followed by the CDTire_MSolve_RS, if your model demands tire resizing.

Version Up for Siemens MF-Tyre/MF-Swift and Cosin FTire (Vehicle Tools Extension)
The Vehicle Tools Extension has upgraded the support for Siemens MF-Tyre/MF-Swift to v2406 and Cosin FTire to 2024-3. These new versions are backward compatible with tire and road input files from previous versions.

Motion-ratio Outputs for Car/Small Truck and Two-Wheeler Library Models (Vehicle Tools Extension)
The Vehicle Tools Extension includes an inbuilt output for reporting motion-ratios of ride springs for the models that are built using the Car/Small truck and Two-wheeler libraries. The calculations are based on the spring displacements and the corresponding wheel center displacements. When you build the models using the libraries, these outputs will be created in dedicated Systems alongside the springs whose motion-ratios are measured.

*Added Contact Jacobian Computation Time to *.`cof` File (MotionSolve)*
When requested, MotionSolve outputs a *.`cof` file summarizing the contact simulation. For each contact, it provides contact details, including the time spent on contact evaluation. In this release, the *.`cof` file is further enhanced to include the computation time for the contact Jacobian as well.

MODE and GETMOD Returns 35 for Assembly Analysis (MotionSolve)
The solver functions MODE and GETMODE provide insight into the current analysis mode of the solver. Both functions have been enhanced so that MotionSolve now returns a value of 35 during Assembly analysis, allowing for clearer differentiation between Assembly and Static analysis modes.

Resolved Issues

- Run time error is encountered after activating joint compliance.
- The **File** > **Open** dialog goes behind the Leaf Spring Builder tool in Linux.

- Output on Entity Sets within a system results in requests for all entities in the model.
- Constraint Mate is not supported with msolve Python.
- The Advanced Selection dialog cannot be invoked from collectors within the FrequencyInput Property Editor.
- Adding a Bushing attachment to a system results in an application error.
- The application crashes when solver variables are added to solver arrays in Linux.
- Entities lose their labels in the Model Browser after undoing the paste action in a system.
- Creating a second Belt pulley system (Substructured flexbodies) takes too long.
- mdl_batch exports an incomplete model from the Vehicle Tools in Linux.
- A force created on a marker has a missing reference after creation.
- The topology view on systems is unreadable when it contains a non-connected entity.
- FMU exported from MotionView is not readable in HyperStudy.
- Y and Z axes in Spline 3D are swapped in the display.
- Zero crossing sensor hmax related to contact is always written despite its selection.
- Updated file paths in FMU parameters are not updated in the exported XML.
- The Contact Force plot is missing in the default Contact report.
- Opening two optimization models consecutively crashes MotionView.
- Datasets with the same variable name have the values synchronized.
- Initial Conditions on systems input through the Entity Editor are not applied to the model.
- Numerical Jacobian calculation for flexible body.
- Functions QDOT and QDDOT must use an actual mode number instead of an index.
- Crash when writing the load set file when the flexible body is not used in any contact.
- Differences in EDEM co-simulation results: Difference between starting the coupling server in Creator Mode and Simulator Mode.
- `edem.fmu` randomly fails to load during the simulation start.
- Model with UserGra causes CNAME to display on all requests in the ABF file.
- mspost now displays the correct progress during ABF export.
- MotionSolve continues saving the XML even if the *savesub* failed.
- MotionSolve supports both label and comment for *post_request* in XML.
- MotionSolve writes only the converged results at the start time of a kinematic analysis to MRF.
- Models with softsoil road graphics created using RoadTools in v2024.0 and v2024.1 are failing.
- CDTire models fail when only one license for CDTire Resizing is available.

  Note: The licensing logic has been changed. Please see *Enhancements* for more details.

- Anti-dive calculations do not report correct values.
- Models with CDTire and FTire crash the solver if the initial static is not included.
- Two-wheeler models with a leaning driver fail after traveling a certain distance for specific event parameters, especially in a Constant radius cornering event.

Known Issues

The following known issues will be addressed in a future release as we continuously improve the software:

- Accessing custom wizards crashes MotionView.

Altair Multiscale Designer 2025 Release Notes

Highlights

- Single Scale Material Support for Forward and Inverse Characterization
- Support of a generic user material class

New Features

User-Defined Material Class

The user-defined material class allows the combination of any elastic law with any supported inelastic or damage law. It can be used to define any needed single scale or constituent material behavior, which is not covered be the predefined material classes (polymer, fiber, tow or metal). Supported combinations are:

Elastic Symmetry	Inelastic Law
Isotropic	Isotropic bilinear brittle damage
	Isotropic 3-piecewise brittle damage
	Orthotropic bilinear brittle damage
	Orthotropic 3-piecewise brittle damage
	Rate independent plasticity
	Rate dependent plasticity
	Hybrid rate independent plasticity and isotropic damage
	Hybrid rate independent plasticity and orthotropic damage
Trans-isotropic	Isotropic bilinear brittle damage
	Isotropic 3-piecewise brittle damage
	Orthotropic bilinear brittle damage
	Orthotropic 3-piecewise brittle damage
Orthotropic	Isotropic bilinear brittle damage
	Isotropic 3-piecewise brittle damage

	Orthotropic bilinear brittle damage
	Orthotropic 3-piecewise brittle damage

Single Scale Material Profile

The Single Scale Material Profile supports forward characterization as well as inverse characterization processes.

All types of material classes (user-defined, polymer, fiber, tow and metal) are supported in the new Single Scale Material Profile.

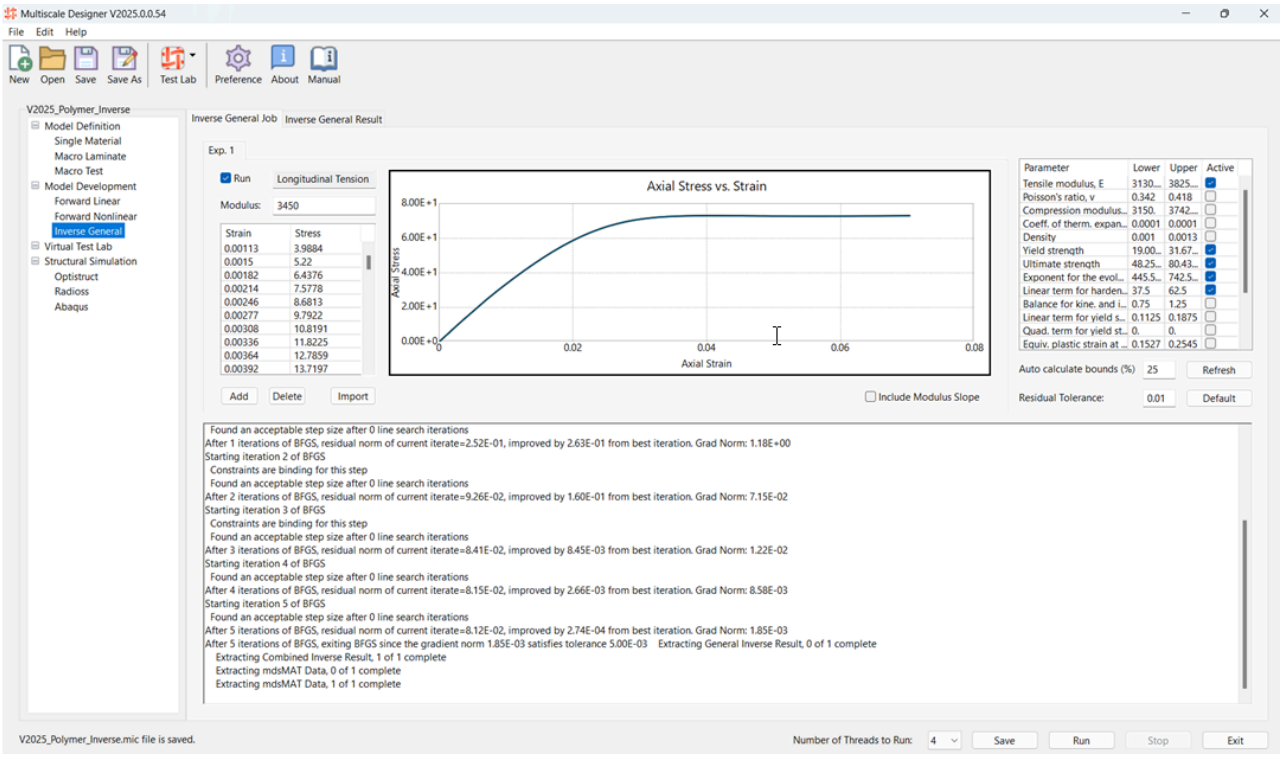

Figure 76: Single Scale Inverse: Input Definition

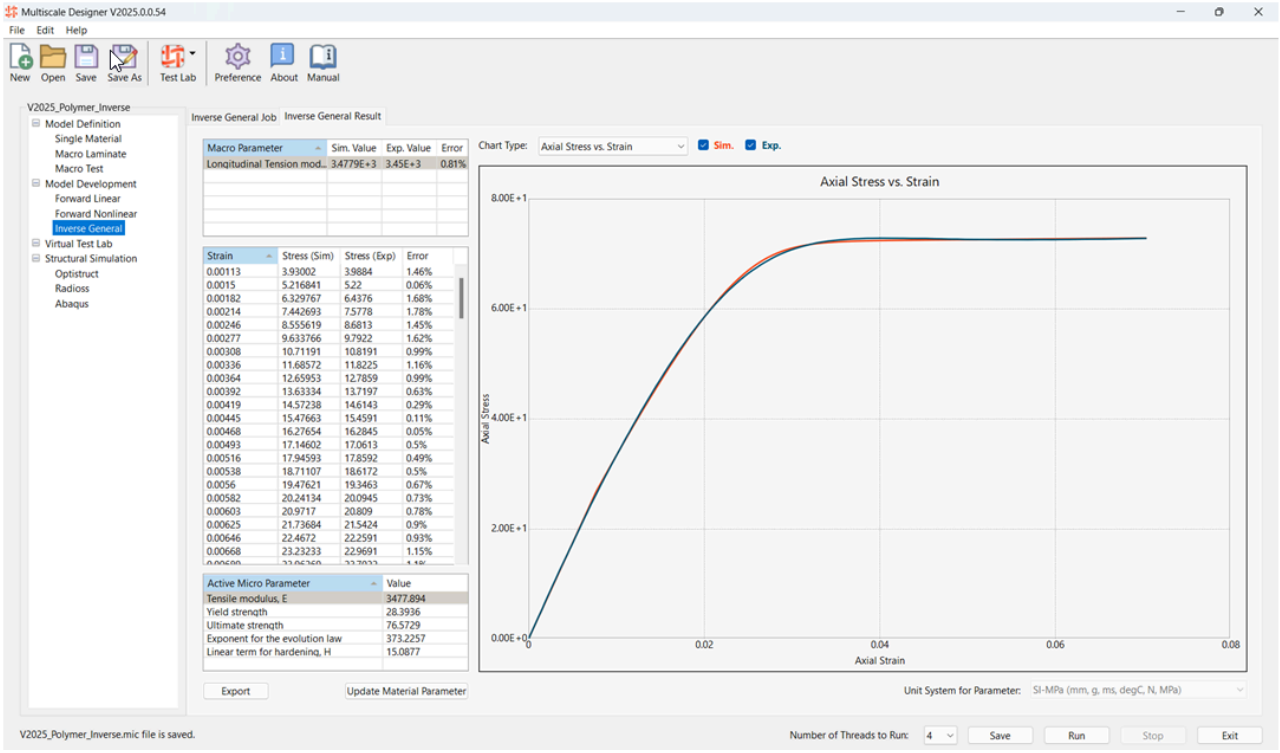

Figure 77: Single Scale Inverse: Result Review

Altair HyperXtrude 2025 Release Notes

Altair HyperXtrude is a suite of finite element solvers for simulating the following manufacturing processes. These solutions have interfaces in Inspire; the HyperXtrude solver is also called the Inspire Extrude Solver.

- Binder Jet Sintering
- Metal Extrusion
- Polymer Extrusion
- Quenching
- Calibration
- Metal Rolling
- Friction Stir Welding
- Resin Transfer Molding

Highlights

- Elastoplastic quenching warpage analysis
- Nose cone animation for transient nose cone analysis

Metal Extrusion

New Features

Nose cone animation

The nose cone emerging from the die is now animated in the bearing and profile 3D regions. This animation is restricted to ram acceleration time. This feature is supported in the transient nose cone and transient 1-cycle analyses. (SLVHXT-777)

Tracking billet skin in multicycle analysis

In a multicycle analysis, the billet skin material migrated into the non-billet regions is now retained in the previous cycle instead of reinitializing the skin data everywhere. This approach allows for simulation of the effect of accumulated skin material in the scrap prediction. (SLVHXT-769, SLVHXT-349)

Enhancements

Parameter to control core heat transfer

A new parameter is added to control the core heat transfer in the mandrel/interior regions of the die. This factor can be set to one of the following three values:

-1 (Default)	The solver picks what is appropriate for the model.
0	The core heat transfer control is off. No special rules are used in the core region. This is more appropriate for solid-profile dies.
1	The core heat transfer control is on. This is more appropriate for hollow-profile dies that have portholes.

If the value is set to 0 or 1, the solver honors this and the value is not internally changed. Using this parameter to turn off the core heat transfer control has improved the profile nose-cone prediction and temperature variation for a few simulations. (SLVHXT-776)

Resolved Issues

Improved accuracy in gap thickness calculation via DCT algorithm

The predicted nose cone depends on the bearing friction, which in turn depends on the thickness computation. In this release, the DCT algorithm is improved to avoid failures under some conditions. With this improvement, thickness computation and nose cone prediction improved for some simulations. (SLVHXT-788)

Flow in the bearing region does not conform to the bearing curve

There was a bug related to the profile shape factor usage in the bearing region's flow computations. With this issue now fixed, the flow in the bearing region is improved. This has resulted in an improved profile nose cone for many simulations. (SLVHXT-773, SLVHXT-634, SLVHXT-736, SLVHXT-719, SLVHXT-755)

Error in writing temperature result in the particle trace CSV file

The temperature is now written for the particles in the CSV file. (SLVHXT-774)

Stat file does not have delimiter specified correctly
This issue was noticed for simulations having multiple exits. The comma is now printed after the exit temperature, therefore for multi-hole models, the data after that no longer appears as a single column. (SLVHXT-775)

Quenching

New Features

Elastoplastic warpage analysis
The quenching simulation now supports elastoplastic analysis. The greatest benefit of this analysis is the prediction of plastic strain that remains in the quenched profile at the end of the simulation. The solver uses the non-uniform temperature distribution during the quenching process to compute the thermal residual stress. In the final step, the profile is cooled to room temperature with minimal constraints using the inertia relief conditions, which retains only the warpage deformation. (SLVHXT-772)

Residual stress in the profile after quenching
The solver computes and uses the thermal stresses for warpage prediction. When the elastoplastic analysis is performed, the stresses are in the final step, which has minimal constraints and no external loading conditions and so can be considered as the residual stresses that cause the predicted warpage. (SLVHXT-716)

Quenching animation control with distance function
A distance function is implemented in the solver to improve the presentation of results correctly in Inspire. This helps ensure Inspire shows only the profile that has exited the die. (SLVHXT-777)

Altair Manufacturing Solver 2025 Release Notes

Please refer to the respective Inspire manufacturing products for the Release Notes.

Altair CFD 2025 Release Notes

Altair AcuSolve 2025 Release Notes

Highlights

- Porous media support for AcuSolve-EDEM coupling.
- Support for multiple field interaction models in AcuSolve-EDEM coupling with multi-drag model selection.

New Features

Porous media support for AcuSolve-EDEM coupling

Modeling a screen or a filter using the AcuSolve-EDEM coupling approach presents a challenge due to the extremely high mesh count required for AcuSolve. Simplification is necessary to reduce computational time. In this scenario, the pressure drop in the flow is modeled using a porous medium, while particle interaction with the screen is handled in EDEM. The *porous_medium_region* parameter in the ELEMENT_SET can be used to activate the porous medium based on the physical velocity porous approach in one of the carrier field elements.

Multiple field interaction model support for EDEM coupling multi drag model selection

In AcuSolve-EDEM coupling, you may need to select different drag models for particles based on their size and shape. To achieve this, each drag model must be defined using the dedicated FIELD_INTERACTION_MODEL. To support this, a new parameter, *edem_particles_names*, is introduced which retrieves the list of EDEM particles referenced in the FIELD_INTERACTION_MODEL.

Coarse graining for EDEM coupling

In AcuSolve-EDEM coupling, the number of fine particles significantly impacts simulation runtime. To reduce computational cost, groups of fine particles can be represented by larger particles, known as "grains," thereby decreasing the total particle count in the simulation. The *edem_coarse_grain_factor* is used to adjust the coupling forces within this coarse-grain system. This factor ensures that the scaled dynamics of the grains accurately represent those of the original fine particles. The coarse grain factor is defined as the ratio of the grain radius to that of an individual original particle.

Non-zero starting time support

A new parameter, *initial_time*, under AUTO_SOLUTION_STRATEGY now allows you to set a non-zero starting time. While available for all standard AcuSolve simulations, this feature is especially useful for AcuSolve-EDEM coupling simulations, enabling AcuSolve to start at a specified non-zero time after initial particles are generated within EDEM.

Enhancements

Enhancements to auto_wall_surface and Improved Result Consistency

auto_wall_surface_output was revisited to streamline and enhance the heat balance process by eliminating duplicate surfaces. This improvement ensures complete and unique auto_wall

surfaces, facilitating accurate heat balance calculations. Additionally, by changing the solver's default setting of exchange_sort_receive better consistency was achieved in results across and run-to-run variations.

Eliminating particle inputs for AcuSolve-EDEM coupling
In previous versions, AcuSolve-EDEM coupling for non-spherical particles required input parameters for sphericity, aspect ratio, and volume using *edem_particles_sphericity_values* and *edem_particles_aspect_ratio_values*. As of this version, these parameter requirements have been eliminated.

Floating monitor surface extended to mesh motion
Starting in version 2023, the floating surface output feature enables output data generation without requiring mesh construction around the desired surface. This version further extends this capability to domains with applied mesh motion.

Time-base curve-fit option for GRAVITY interpolation
In the GRAVITY command, a new curve fit variable of time is introduced, enabling you to easily interpolate user data over time without needing to rely on UDFs.

acuOdb enhancements
You can now export AcuSolve data in Abaqus (.odb) format with enhanced options: You can specify a particular element set using the '-outs' option and apply model scaling with the '-scale' parameter.

Documentation Additions
Training manual

- The thermal-electric battery simulation setup and guidelines in the Solver Feature Guidelines have been updated, summarizing the required battery module input parameters and adding a new section detailing ECM parameter specifications for the battery model.
- A new section on submitting AcuSolve jobs on Linux servers has been added to the Solver Feature Guidelines.
- Details about the Eulerian multiphase models and turbulence models are now available under the AcuSolve Solver Features section.

SimLab-based Tutorial Updates
Two (2) tutorials for the SimLab CFD user interface have been updated. The updated tutorials are:

- ACU-T: 3513/SL-2176 ARC-based Thermal Runaway: 1S1P Module
- ACU-T: 7202/SL-2502: CFD Topology Optimization of Multiple Flow Paths

Resolved Issues

- Users reported that acuSig for clean-stop was not functioning properly, with issues depending on the Linux version and hardware. This has been resolved in the 2025 version.

Altair nanoFluidX 2025 Release Notes

Highlights

- Up to 1.2x performance improvement in Tartakovsky F1 surface tension model.

New Features

Optional reference velocity in body frame

Specifying reference velocity is now optional when body frame is active. This simplifies case setup where high acceleration events are more dominant than high speed events, for example in predominantly vibrational tank sloshing cases.

Independent interpolation domain size in nFX[c]

Interpolation domain size of nFX[c] can now be set independently from nanoFluidX. This simplifies narrowing down the post-processing data to the volume of interest.

Enhancements

Performance improvement in Tartakovsky F1 surface tension model

Tartakovsky F1 surface tension model implementation has been modified, offering up to 1.2x speedup compared to prior implementation.

Load balancing adjustment for cases with large wetting area

Load balancing method in Weighted interaction scheme has been adjusted to account for cases with larger wetted surface to fluid bulk ratio, offering performance improvements for some water management and water wading cases.

Memory usage summary

In addition to the running memory usage, a summary of peak memory usage of host and device can now be shown at the end of the simulation log. The summary is activated via specifying relevant options in the configuration file.

Availability of cell mesh information prior to simulation

Domain addressing and cell mesh data can now be viewed when running nFX[p], nFX[c] in --nfxp mode or Data Check in SimLab.

Resolved Issues

- Total number of interpolation points exceeding 2147483647 led to unexpected behavior in nFX[c].

Altair ultraFluidX 2025 Release Notes

New Features

Introduction of additional output variables

ultraFluidX 2025 directly outputs the standard deviation (STD) and variance of pressure enabling advanced and fast analysis of flow statistics. Additionally, you now have the option to output velocity magnitude instead of, or in addition to, the velocity components. Velocity magnitude output in place of the velocity components lead to a reduction of disk space by approximately 66 percent.

Coarsening of outputs

Prior to this release, displaying volumetric quantities such as 3D streamlines and iso-surfaces required large amount of disk space and significant time to post-process, particularly for transient and production level datasets. To address this, ultraFluidX 2025 allows output to be written at a

coarser resolution than that used during simulation. This approach preserves the accurately of the computations while reducing output file sizes by aggregating multiple voxel values into a single, coarser voxel. Using a refinement-based strategy, one level of coarsening is available, 2x2x2. Assuming a region space discretized with a single resolution, a 2x2x2 coarsening would lead to a reduction of the output data size by approximatively 8x.

Enhancements

Output bounding boxes

ultraFluidX 2025 enhances output versatility by introducing bounding boxes for full output and/or selected surfaces. This feature, building on the existing functionality for partial volumes, enables you to define specific regions of interest within the flow field for more targeted data export. You can now write one or multiple subsets of volume and surface data, concentrating on the most critical areas of analysis, which improves both data relevance and storage efficiency.

Reducing export time and memory requirements for writing H3D files

ultraFluidX 2025 introduces an improved H3D output files algorithm resulting in significantly shorter export times. Additionally, memory usage on the calculation node for merging data across all ranks into a single file has been substantially reduced, enhancing data export performance and efficiency. These improvements are particularly beneficial for Computational Aeroacoustics (CAA) solutions, where managing large datasets is essential for accurately capturing transient and acoustic phenomena.

Extension of the virtual fan model

In ultraFluidX 2025, the virtual fan model, used to simulate fan effects on the flow field via a source region, has been enhanced to include the pressure loss (resistance) induced by the fan. Additionally, a new method has been introduced for modeling the tangential velocity. Alongside the existing Hough-Ordway method, you now have the option to employ a linear relation, linking circumferential speed to rotational speed and position within the fan plane. These updates provide greater flexibility and precision for modeling fan-driven airflow dynamics in complex simulations.

Altair HyperView Player 2025 Release Notes

Resolved Issues

Application error when turning on/off Axisymmetry from Display Control.

Altair Multibody Solutions 2025 Release Notes

Highlights

The 2025 release of Altair Multibody Solutions introduces several enhancements and bug fixes to MotionView, MotionSolve, and their extensions. Here are some of the key highlights:

- Save system definition to the Entity Browser (MotionView)
- Deformable curve on a Polybeam (MotionView)
- Improvements to Frequency Response Analysis (MotionSolve)
- CD Tire updates (MotionSolve/Vehicle Tools Extension)

New Features

Save System Definition to the Entity Browser (MotionView)

MotionView's Entity Browser contains a list of entities that can be added to the model. It is now possible to save systems to the Entity Browser for quick retrieval and reuse.

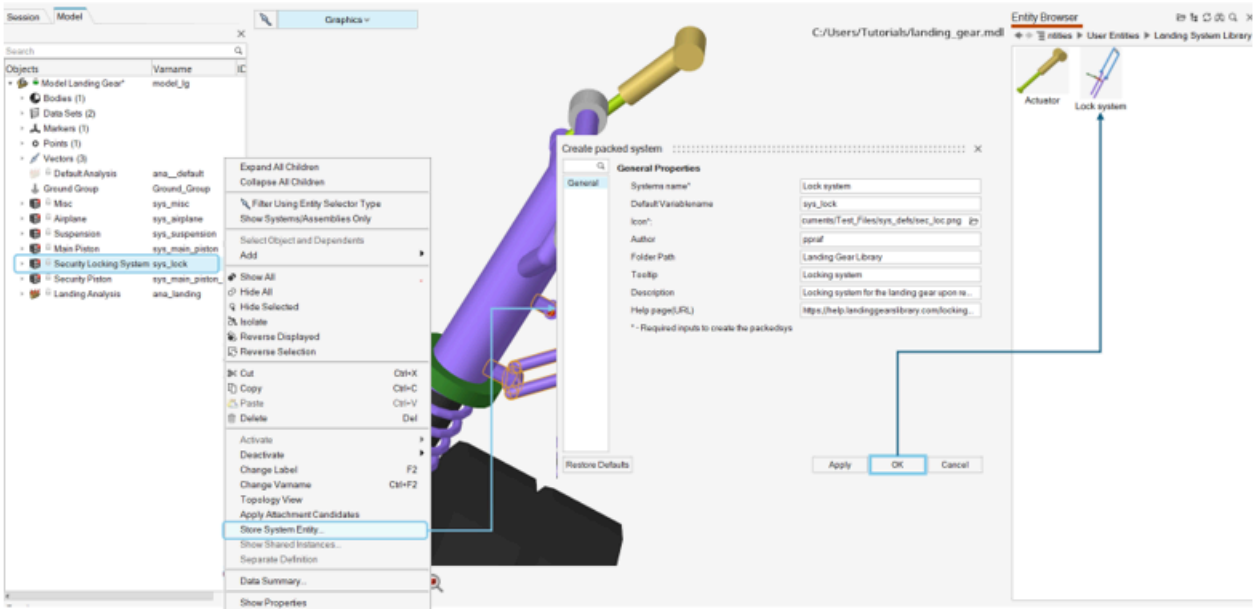

Figure 78:

Select a system in the Model Browser, right-click, and select **Store System Entity...**. In the subsequent dialog, fill in system details, an icon image, and click **OK**.

- The system is listed under the User Entities folder.
- Provide a folder path to organize the stored systems.
- Double click on the stored system to insert it into the active model.

Deformable Curve on Polybeam (MotionView)

In the previous release, MotionView and MotionSolve added support for the deformable curve contact, which can be used to model pipes, cables, and tubes in contact. In this release, MotionView further enhances this functionality by providing an option to add a deformable curve

with graphics when creating a polybeam entity. This improvement streamlines the setup of models with slender elements, enabling faster configuration.

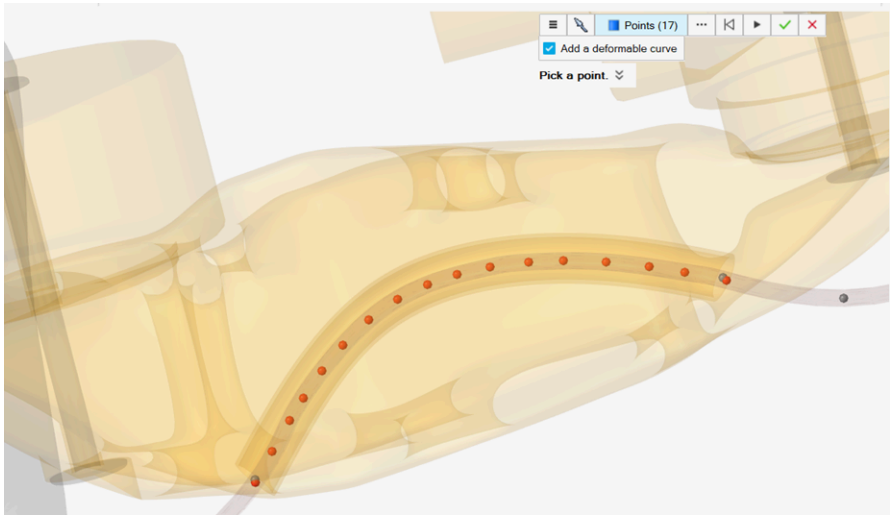

Figure 79:

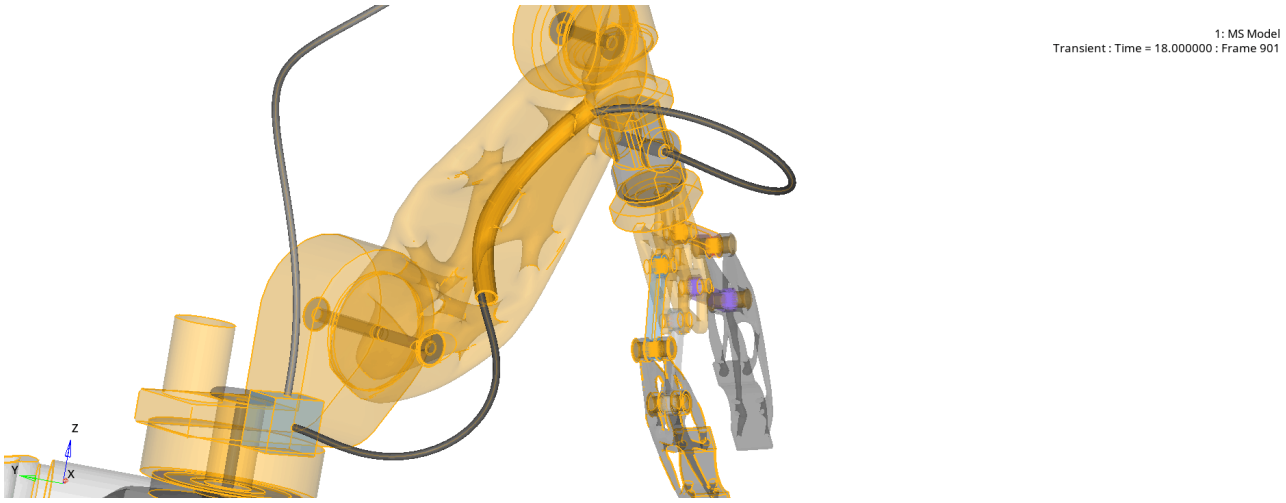

Figure 80:

Enhancements

Import Markers and Geometrical Curves from Catia (MotionView)

The CAD import functionality has been enhanced to read any existing marker and geometrical curve from Catia V5.

Improvements to Frequency Response Analysis (MotionSolve)

In this release, the Frequency Response Analysis in MotionSolve introduces additional support for flexible bodies:

- Frequency-dependent modal participation factors.
- Frequency-dependent kinetic energy, dissipative energy, and strain energy distribution.
- Frequency-dependent transfer path analysis.

Write mkb Matrix in Linear Analysis (MotionSolve)
In this release, MotionSolve includes support for exporting of the mass, stiffness, and damping matrices of a linearized system.

Fraunhofer CD-Tire: Resizing (Vehicle Tools Extension)
The Vehicle Tools Extension now supports the Resizing feature for CDTire from Fraunhofer ITWM. The resizing feature allows a tire model to be automatically resized based on reference and target tire and rim specifications.

Fraunhofer CD-Tire: Support for MF++ (Vehicle Tools Extension)
The Vehicle Tools Extension now supports the Fraunhofer MF++ tires. CDTire/MF++ is a temperature enhanced Magic Formula for coupling to CDTire/Thermal in advanced handling applications.

Fraunhofer CD-Tire: Licensing Changes (MotionSolve)
MotionSolve has updated the license check-out logic for CDTire and the CDTire resizing feature. MotionSolve supports the usage of the CD-Tire resizing feature. To use this feature, your Altair license must include both the CDTire_MSolve (regular CD Tire) and CDTire_MSolve_RS (CD Tire with resizing) features. MotionSolve first checks out the CDTire_MSolve feature, followed by the CDTire_MSolve_RS, if your model demands tire resizing.

Version Up for Siemens MF-Tyre/MF-Swift and Cosin FTire (Vehicle Tools Extension)
The Vehicle Tools Extension has upgraded the support for Siemens MF-Tyre/MF-Swift to v2406 and Cosin FTire to 2024-3. These new versions are backward compatible with tire and road input files from previous versions.

Motion-ratio Outputs for Car/Small Truck and Two-Wheeler Library Models (Vehicle Tools Extension)
The Vehicle Tools Extension includes an inbuilt output for reporting motion-ratios of ride springs for the models that are built using the Car/Small truck and Two-wheeler libraries. The calculations are based on the spring displacements and the corresponding wheel center displacements. When you build the models using the libraries, these outputs will be created in dedicated Systems alongside the springs whose motion-ratios are measured.

*Added Contact Jacobian Computation Time to *.`cof` File (MotionSolve)*
When requested, MotionSolve outputs a *.`cof` file summarizing the contact simulation. For each contact, it provides contact details, including the time spent on contact evaluation. In this release, the *.`cof` file is further enhanced to include the computation time for the contact Jacobian as well.

MODE and GETMOD Returns 35 for Assembly Analysis (MotionSolve)
The solver functions MODE and GETMODE provide insight into the current analysis mode of the solver. Both functions have been enhanced so that MotionSolve now returns a value of 35 during Assembly analysis, allowing for clearer differentiation between Assembly and Static analysis modes.

Resolved Issues

- Run time error is encountered after activating joint compliance.
- The **File** > **Open** dialog goes behind the Leaf Spring Builder tool in Linux.
- Output on Entity Sets within a system results in requests for all entities in the model.
- Constraint Mate is not supported with msolve Python.

- The Advanced Selection dialog cannot be invoked from collectors within the FrequencyInput Property Editor.
- Adding a Bushing attachment to a system results in an application error.
- The application crashes when solver variables are added to solver arrays in Linux.
- Entities lose their labels in the Model Browser after undoing the paste action in a system.
- Creating a second Belt pulley system (Substructured flexbodies) takes too long.
- mdl_batch exports an incomplete model from the Vehicle Tools in Linux.
- A force created on a marker has a missing reference after creation.
- The topology view on systems is unreadable when it contains a non-connected entity.
- FMU exported from MotionView is not readable in HyperStudy.
- Y and Z axes in Spline 3D are swapped in the display.
- Zero crossing sensor hmax related to contact is always written despite its selection.
- Updated file paths in FMU parameters are not updated in the exported XML.
- The Contact Force plot is missing in the default Contact report.
- Opening two optimization models consecutively crashes MotionView.
- Datasets with the same variable name have the values synchronized.
- Initial Conditions on systems input through the Entity Editor are not applied to the model.
- Numerical Jacobian calculation for flexible body.
- Functions QDOT and QDDOT must use an actual mode number instead of an index.
- Crash when writing the load set file when the flexible body is not used in any contact.
- Differences in EDEM co-simulation results: Difference between starting the coupling server in Creator Mode and Simulator Mode.
- `edem.fmu` randomly fails to load during the simulation start.
- Model with UserGra causes CNAME to display on all requests in the ABF file.
- mspost now displays the correct progress during ABF export.
- MotionSolve continues saving the XML even if the *savesub* failed.
- MotionSolve supports both label and comment for *post_request* in XML.
- MotionSolve writes only the converged results at the start time of a kinematic analysis to MRF.
- Models with softsoil road graphics created using RoadTools in v2024.0 and v2024.1 are failing.
- CDTire models fail when only one license for CDTire Resizing is available.

  Note: The licensing logic has been changed. Please see *Enhancements* for more details.

- Anti-dive calculations do not report correct values.
- Models with CDTire and FTire crash the solver if the initial static is not included.
- Two-wheeler models with a leaning driver fail after traveling a certain distance for specific event parameters, especially in a Constant radius cornering event.

Known Issues

The following known issues will be addressed in a future release as we continuously improve the software:

- Accessing custom wizards crashes MotionView.

Altair SEAM 2025 Release Notes

Highlights

- Independent graphics behavior for SEA entities.

New Features

Independent Graphics Behavior for SEA Entity Files

A new feature has been added for independent visualization for all SEA Entities: Subsystems, Junctions, NCTs, and Excitations. With this feature, the selection and handling of SEA entities is comparable to FEA entities, which improves the usability of the SEA entities.

Enhancements

Excitation Icons

New icons have been added.

Resolved Issues

- Normal visualization of SEA entities.
- Saving Stiffness Multiplier OUT files.

Altair HyperWorks Licensing 2025 Release Notes

Highlights

- Altair License Manager 2025 (and its updates) is required for Altair 2025 and its updates.
- HyperWorks 9.1, 10.1, 11.0, 12.0, 13.0, 14.0, 2017, 2018, 2019, 2020, 2021, 2022, 2023, 2024 as well as Altair 2025 are compatible with Altair License Manager 2025.

New Features

Product License Features

The following Altair HyperWorks applications/tools are available under Altair Units (AUs) licensing:

Battery Damage Identifier

Battery Damage Identifier can predict the behavior of mechanically damaged batteries and modules, focusing on the modeling of the mechanical and electrothermal properties of its components. A multi-physics process was designed to identify the mechanical and electrothermal properties of the homogenized material, based on the detailed composition of the cell. It contains tools to evaluate adiabatic or quasi-static temperature distribution in damaged cells. This feature requires 50 AUs.

Altair Copilot AI Assistant for HyperWorks

AI-powered support assistant that understands natural language and can answer questions based exclusively on Altair's online help material and other Altair knowledge sources. It takes 2 AUs for this feature.

DSim

DSim is a digital simulator and electronic functional verification application. DSim Solver license draws units on a per-CPU core basis. The unit draw is determined by the total number of cores for all concurrent solver instances according to a lookup table.

Number of Cores	Draw per Core
1-128	10
> 129	5

The free version of DSim Desktop for single core usage continues to exist, still using the legacy licensing. This will be moved to Altair feature based licensing after the 2025 release.

PSIM Solver

PSIMSolver is the new license feature introduced in 2025 for the Solver in PSIM. When a simulation is executed either from within the PSIM GUI or from PSIM Solver (standalone), the PSIMSolver license feature is required. For Solver InApp licensing: PSIMSolver takes 15 AUs with Decay factor for multiple job runs. For Solver HPC licensing: the unit draw is on a per-CPU core basis. The unit draw is determined by the total number of cores for a solver according to a lookup table.

SimLab

SimLab v2025 introduces license feature, SimLabEDEMSolver. This feature is required when running EDEM solver in SimLab for one core. Multi-core usage follows the Solver In-App Licensing table.

SimLab

SimLab v2025 introduces license feature, SimLabFekoSolver. This feature is required when running Feko solver in SimLab for one core. Multi-core usage follows the Solver In-App Licensing table.

Licensing Changes

Product Licensing Changes

The following products have made licensing changes:

Engineering Data Sciences

The license feature, EngDataSciPaiShape, is used for Prediction of simulation results without requiring parametric inputs. Altair physicsAI model training requires 75 AUs. Altair physicsAI models require 75 AUs for inference within Altair modeling and visualization products, otherwise they require 150 AUs.

romAI

The license feature, romAI, that is required within Twin Activate to leverage ROMs for computationally expensive simulations draws 60 AUs.
The ActivateLicFreeExport feature is available on Altair Managed License only.

FlightStream

The unit draws for FlightStream will no longer scale with the number of cores used. Draws will now be 50 AUs flat per instance of the application.

Altair Software and Hardware 2025 Release Notes

Hardware and Platform Support

Operating system additions

Support for the following platforms has been added:

- Windows 11
- RHEL/CIQ Rocky Linux 9.4

Discontinued hardware platform support systems

Support for the following platforms has been dropped:

- SLES 15 SP3
- RHEL 8.4

Installation

HyperWorks 2025 suite contains independent product packages

Independent product packages give you the flexibility to download desired packages as needed. Any HyperWorks 2025 SA (standalone) update package can be installed either on top of an existing 2025 installation or it can be installed independently without the prerequisite of a 2025 base.

> **Important:** You can no longer install a different version of an already installed product to the same location (for example, HW Desktop 2025.1 on top of HW Desktop 2025).

Installer capabilities and features

HyperWorks 2025 brings with it the robust capabilities and features of HyperWorks 2024.1 installers, as well as additional support for HotFix installers. These HotFix installers are quick patch type installers that are used to install on top of a prerequisite base installation, unlike the other full installers. These are marked with the word “hotfix” in the installer name and bring with them the quick fixes and features to better suit your needs.

Model files removed from installation package

To reduce the footprint size, the following tutorial model and demo model files are no longer included in the local installation. You can now find zipped tutorial model files and demo model files on Altair One via the Altair One Community and Altair One Marketplace sites. Altair recommends that you create an Altair One account and use it as your primary portal to access product documentation, a Knowledge Base, and customer support.

- Tutorial Model Files
  - BasicFEA
  - HyperGraph
  - HyperLife
  - HyperLife Weld Certification
  - HyperMesh

    - HyperStudy
    - HyperView
    - HyperWorks
    - HyperWorks CFD
    - HyperWorks Collaboration Tools
    - HyperWorks Desktop
    - MotionSolve
    - MotionView
    - nanoFluidX
    - OptiStruct
    - Radioss
    - SnRD
- Demo Model Files
    - OptiStruct > Examples
    - OptiStruct > Verification Problems
    - Radioss > Examples
    - Radioss > Verification Problems

Installation and Packaging Features

- Application Java Runtime was upgraded to Amazon Corretto 11.0.24.8.1.
- The installer JRE was updated to Amazon Corretto 11.0.19.7.1.
- A launch script was added for HyperLife Crack Growth on Linux.
- Inspire products and HyperMesh CFD now have their own lightweight installers that will download and install all required components.
- HyperMesh CFD now supports server-mode installation.
- Altair Inspire was added to the Linux menu.
- HyperStudy has been updated with a new user interface.
- Desktop 2022.3 can be installed using the 2025 Altair HyperWorks Products Installer (main installer).
- HyperMesh CFD was added as a new product on Windows and Linux.
- Installers were upgraded to InstallAnywhere 2022 R2.
- Digital signatures have been added to more executables to improve security.
- Added HyperWorks Products Installer support for Inspire on Linux.
- Installers have been improved to reduce the file footprint and improve installation performance.
- Mechanical Solvers installation is now modular, so you can choose which solvers to install.
- nanoFluidX was added to the CFD Solvers package on Windows, it is also available on Linux.
- File associations for HyperWorks Desktop were updated to open files in the new user interface.
- FluxMotor was added to the Flux package on Windows.
- newFASANT was added to the Feko package on Windows and Linux.

- Overwrite/overlay installation is no longer supported, and the installers will prevent it from happening.

Known Issues

The following known issues will be addressed in a future release as performance of the software is continuously improved:

- The use of commas in the install path causes multiple products to not function currently.
- On Windows 11, the shortcut icons for server and client mode installations are blank unless remote paths in shortcut icons is enabled in the Windows group policy editor.
- Using the Back/Previous option with an installer may not reset the installer variables correctly if changes are made and the install session is continued.
- On Linux, installing to a directory with spaces is not supported. Additionally, if the path that the installers are run from contains a space, you will see a harmless message in the terminal window.
- Special characters in the path to the temp location can cause the Java in the installer to not function correctly.
- On Linux, during a console mode install, when using the response file output command `-r` in the Help Configuration response file may not write out correctly.
- If there is not enough TMP/TEMP space on the main drive (for example, `C:` under Windows), the installer will prompt you to input a new location. Some Java TMP/TEMP space will still try to go to the main drive even when selecting this new location.
- Click **Cancel** to stop an installation:
  - via the Altair HyperWorks Products Installer (main installer) does not clean up the already installed files, but does clean up the files in the TMP/TEMP folder. You will need to uninstall/delete the installed files from the chosen install folder manually if **Cancel** was clicked.
  - causes the Add/Remove uninstaller icon to possibly not function correctly, and then manual removal of the installed files would be necessary.
- The HyperWorks installers do not support the use of the default `InstallAnywhere installer.properties` files. It is requested that a unique name is used when using a response file; for example, `installer_hw2024.properties`, and to use it with the `-f <path_to_file>/<filename>` command line option.
- For NETSETUP client installs from a server installation of HyperWorks, a UNC path must be used to get to the server installation path through Windows Explorer or Windows Network.
- On Linux, there is an issue with using the Altair HyperWorks Products Installer (main installer) with the IATEMPDIR variable to specify a custom temporary directory location. All sub-installers will try to use `/tmp` from the machine running the installers for temporary installer files.
- The installer does not run the Linux Menu installation by default, the user must run the installation script.
- When the Main Uninstaller is used, desktop shortcuts, if any, will not be removed during uninstallation.

Resolved Issues

- Apache Tomcat security vulnerability was addressed in the PBS Access Desktop (PAD) common installer.

- Installers that were failing to install when the Windows Regional Format was set to Arabic or Hebrew now install successfully.
- Running the NETSETUP client installer as a non-administrative user now works properly.
- EDEM will no longer set file associations if you have not selected that option when using the HyperWorks Products Installer.
- Hydra services started by Feko and Flux were not being stopped before an uninstall.
- The installer names were being truncated on displays with scaling set to 125% or higher.
- Installing Feko as a non-administrator no longer causes a non-fatal installation error.
- Installing SimSolid by itself using the Altair HyperWorks Products installer (main installer) no longer causes a non-fatal installation error.
- Installing Flux in silent mode on Linux no longer causes an unexpected terminal message.
- HyperMesh File Preview (thumbnail) registration was not occurring for some types of installations.
- Silent installs will no longer check for running processes using PowerShell.
- When SimLab is uninstalled from a multi-product install, it no longer removes the common Python directory.
- The Altair HyperWorks Products Installer (main installer) reported an incorrect amount of available disk space on Linux.
- The most recent package was not selected in the Altair HyperWorks Products Installer (main installer) Choose Install Set panel in some cases.
- Multiple successive spaces in the installation directory or shortcut folder caused an issue on Windows.

Intellectual Property Rights Notice

This Intellectual Property Rights Notice is exemplary, and therefore not exhaustive, of the intellectual property rights held by Altair Engineering Inc. or its affiliates. Software, other products, and materials of Altair Engineering Inc. or its affiliates are protected under laws of the United States and laws of other jurisdictions.

In addition to intellectual property rights indicated herein, such software, other products, and materials of Altair Engineering Inc. or its affiliates may be further protected by patents, additional copyrights, additional trademarks, trade secrets, and additional other intellectual property rights. For avoidance of doubt, copyright notice does not imply publication. Copyrights in the below are held by Altair Engineering Inc. or its affiliates. Additionally, all non-Altair marks are the property of their respective owners. If you have any questions regarding trademarks or registrations, please contact marketing and legal.

This Intellectual Property Rights Notice does not give you any right to any product, such as software, or underlying intellectual property rights of Altair Engineering Inc. or its affiliates. Usage, for example, of software of Altair Engineering Inc. or its affiliates is governed by and dependent on a valid license agreement.

Altair® HyperWorks®, a Design & Simulation Platform

Altair® AcuSolve® ©1997-2024

Altair® Activate® ©1989-2024

Altair® Automated Reporting Director™ ©2008-2022

Altair® Battery Damage Identifier™ ©2019-2024

Altair® CFD™ ©1990-2024

Altair Compose® ©2007-2024

Altair® ConnectMe™ ©2014-2024

Altair® DesignAI™ ©2022-2024

Altair® DSim™ ©2024

Altair® EDEM™ ©2005-2024

Altair® EEvision™ ©2018-2024

Altair® ElectroFlo™ ©1992-2024

Altair Embed® ©1989-2024

Altair Embed® SE ©1989-2024

Altair Embed®/Digital Power Designer ©2012-2024

Altair Embed®/eDrives ©2012-2024

Altair Embed® Viewer©1996-2024

Altair® e-Motor Director™©2019-2024

Altair® ESAComp® ©1992-2024

Altair® expertAI™ ©2020-2024

Altair® Feko® ©1999-2024

Altair® FlightStream® ©2017-2024

Altair® Flow Simulator™ ©2016-2024

Altair® Flux® ©1983-2024

Altair® FluxMotor® ©2017-2024

Altair® GateVision PRO™ ©2002-2024

Altair® Geomechanics Director™ ©2011-2022

Altair® HyperCrash® ©2001-2023

Altair® HyperGraph® ©1995-2024

Altair® HyperLife® ©1990-2024

Altair® HyperMesh® ©1990-2024

Altair® HyperMesh® CFD ©1990-2024

Altair® HyperMesh ® NVH ©1990-2024

Altair® HyperSpice™ ©2017-2024

Altair® HyperStudy® ©1999-2024

Altair® HyperView® ©1999-2024

Altair® HyperView Player® ©2022-2024

Altair® HyperWorks® ©1990-2024

Altair® HyperWorks® Design Explorer ©1990-2024

Altair® HyperXtrude® ©1999-2024

Altair® Impact Simulation Director™ ©2010-2022

Altair® Inspire™ ©2009-2024

Altair® Inspire™ Cast ©2011-2024

Altair® Inspire™ Extrude Metal ©1996-2024

Altair® Inspire™ Extrude Polymer ©1996-2024

Altair® Inspire™ Form ©1998-2024

Altair® Inspire™ Mold ©2009-2024

Altair® Inspire™ PolyFoam ©2009-2024

Altair® Inspire™ Print3D ©2021-2024

Altair® Inspire™ Render ©1993-2024

Altair® Inspire™ Studio ©1993-2024

Altair® Material Data Center™ ©2019-2024

Altair® Material Modeler™ ©2019-2024

Altair® Model Mesher Director™ ©2010-2024

Altair® MotionSolve® ©2002-2024

Altair® MotionView® ©1993-2024

Altair® Multi-Disciplinary Optimization Director™ ©2012-2024

Altair® Multiscale Designer® ©2011-2024

Altair® newFASANT™ ©2010-2020

Altair® nanoFluidX® ©2013-2024

Altair® NLVIEW® ©2018-2024

Altair® NVH Director™ ©2010-2024

Altair® NVH Full Vehicle™ ©2022-2024

Altair® NVH Standard™ ©2022-2024

Altair® OmniV™ ©2015-2024

Altair® OptiStruct® ©1996-2024

Altair® physicsAI™ ©2021-2024

Altair® PollEx™ ©2003-2024

Altair® PollEx™ for ECAD ©2003-2024

Altair® PSIM™ ©1994-2024

Altair® Pulse™ ©2020-2024

Altair® Radioss® ©1986-2024

Altair® romAI™ ©2022-2024

Altair® RTLvision PRO™ ©2002-2024

Altair® S-CALC™ ©1995-2024

Altair® S-CONCRETE™ ©1995-2024

Altair® S-FRAME® ©1995-2024

Altair® S-FOUNDATION™ ©1995-2024

Altair® S-LINE™ ©1995-2024

Altair® S-PAD™ ©1995-2024

Altair® S-STEEL™ ©1995-2024

Altair® S-TIMBER™ ©1995-2024

Altair® S-VIEW™ ©1995-2024

Altair® SEAM® ©1985-2024

Altair® shapeAI™ ©2021-2024

Altair® signalAI™ ©2020-2024

Altair® Silicon Debug Tools™ ©2018-2024

Altair® SimLab® ©2004-2024

Altair® SimLab® ST ©2019-2024

Altair® SimSolid® ©2015-2024

Altair® SpiceVision PRO™ ©2002-2024

Altair® Squeak and Rattle Director™ ©2012-2024

Altair® StarVision PRO™ ©2002-2024

Altair® Structural Office™ ©2022-2024

Altair® Sulis™ ©2018-2024

Altair® Twin Activate® ©1989-2024

Altair® UDE™ ©2015-2024

Altair® ultraFluidX® ©2010-2024

Altair® Virtual Gauge Director™ ©2012-2024

Altair® Virtual Wind Tunnel™ ©2012-2024

Altair® Weight Analytics™ ©2013-2022

Altair® Weld Certification Director™ ©2014-2024

Altair® WinProp™ ©2000-2024

Altair® WRAP™ ©1998-2024

Altair® HPCWorks®, a HPC & Cloud Platform

Altair® Allocator™ ©1995-2024

Altair® Access™ ©2008-2024

Altair® Accelerator™ ©1995-2024

Altair® Accelerator™ Plus ©1995-2024

Altair® Breeze™ ©2022-2024

Altair® Cassini™ ©2015-2024

Altair® Control™ ©2008-2024

Altair® Desktop Software Usage Analytics™ (DSUA) ©2022-2024

Altair® FlowTracer™ ©1995-2024

Altair® Grid Engine® ©2001, 2011-2024

Altair® InsightPro™ ©2023-2024

Altair® Hero™ ©1995-2024

Altair® Liquid Scheduling™ ©2023-2024

Altair® Mistral™ ©2022-2024

Altair® Monitor™ ©1995-2024

Altair® NavOps® ©2022-2024

Altair® PBS Professional® ©1994-2024

Altair® PBS Works™ ©2022-2024

Altair® Simulation Cloud Suite (SCS) ©2024

Altair® Software Asset Optimization (SAO) ©2007-2024

Altair® Unlimited™ ©2022-2024

Altair® Unlimited Data Analytics Appliance™ ©2022-2024

Altair® Unlimited Virtual Appliance™ ©2022-2024

Altair® RapidMiner®, a Data Analytics & AI Platform

Altair® AI Hub ©2023-2024

Altair® AI Edge™ ©2023-2024

Altair® AI Cloud ©2022-2024

Altair® AI Studio ©2023-2024

Altair® Analytics Workbench™ ©2002-2024

Altair® Graph Lakehouse™ ©2013-2024

Altair® Graph Studio™ ©2007-2024

Altair® Knowledge Hub™ ©2017-2024

Altair® Knowledge Studio® ©1994-2024

Altair® Knowledge Studio® for Apache Spark ©1994-2024

Altair® Knowledge Seeker™ ©1994-2024

Altair® IoT Studio™ ©2002-2024

Altair® Monarch® ©1996-2024

Altair® Monarch® Classic ©1996-2024

Altair® Monarch® Complete™ ©1996-2024

Altair® Monarch® Data Prep Studio ©2015-2024

Altair® Monarch Server™ ©1996-2024

Altair® Panopticon™ ©2004-2024

Altair® Panopticon™ BI ©2011-2024

Altair® SLC™ ©2002-2024

Altair® SLC Hub™ ©2002-2024

October 7, 2024

Technical Support

Altair's support resources include engaging learning materials, vibrant community forums, intuitive online help resources, how-to guides, and a user-friendly support portal.

Visit Customer Support to learn more about our support offerings.