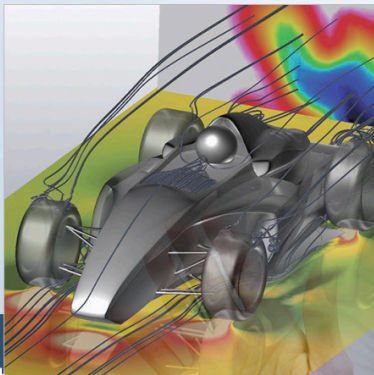
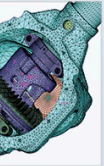
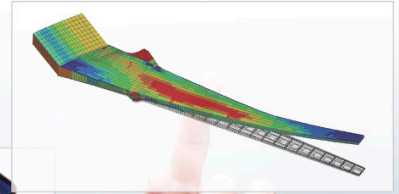
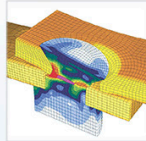




HyperWorks®

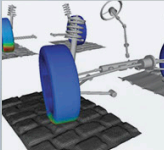
Composites



Crash
and Safety



NVH



Altair Engineering Support Contact Information

Web site www.altairhyperworks.com

Location	Telephone	e-mail
Australia	64.9.413.7981	anzsupport@altair.com
Brazil	55.11.3884.0414	br_support@altair.com
Canada	416.447.6463	support@altairengineering.ca
China	86.400.619.6186	support@altair.com.cn
France	33.1.4133.0992	francesupport@altair.com
Germany	49.7031.6208.22	hwsupport@altair.de
India	91.80. 6629.4500 1.800.425.0234 (toll free)	support@india.altair.com
Italy	39.800.905.595	support@altairengineering.it
Japan	81.3.5396.2881	support@altairjp.co.jp
Korea	82.70.4050.9200	support@altair.co.kr
Mexico	55.56.58.68.08	mx-support@altair.com
New Zealand	64.9.413.7981	anzsupport@altair.com
North America	248.614.2425	hwsupport@altair.com
Scandinavia	46.46.460.2828	support@altair.se
United Kingdom	01926.468.600	support@uk.altair.com

In addition, the following countries have resellers for Altair Engineering: Colombia, Czech Republic, Ecuador, Israel, Russia, Netherlands, Turkey, Poland, Singapore, Vietnam, Indonesia

Official offices with resellers: Canada, China, France, Germany, India, Malaysia, Italy, Japan, Korea, Spain, Taiwan, United Kingdom, USA

Copyright© Altair Engineering Inc. All Rights Reserved for:

HyperMesh® 1990-2014; HyperCrash® 2001-2014; OptiStruct® 1996-2014; RADIOSS®1986-2014; HyperView®1999-2014; HyperView Player® 2001-2014; HyperStudy® 1999-2014; HyperGraph®1995-2014; MotionView® 1993-2014; MotionSolve® 2002-2014; HyperForm® 1998-2014; HyperXtrude® 1999-2014; Process Manager™ 2003-2014; Temples™ 1990-2014; TextView™ 1996-2014; MediaView™ 1999-2014; TableView™ 2013-2014; BatchMesher™ 2003-2014; HyperMath® 2007-2014; Manufacturing Solutions™ 2005-2014; HyperWeld® 2009-2014; HyperMold® 2009-2014; solidThinking® 1993-2014; solidThinking Inspire® 2009-2014; solidThinking Evolve®™ 1993-2014; Durability Director™ 2009-2014; Suspension Director™ 2009-2014; AcuSolve® 1997-2014; AcuConsole® 2006-2014; SimLab®™2004-2014 and Virtual Wind Tunnel™ 2012-2014.

In addition to HyperWorks® trademarks noted above, Display Manager™, Simulation Manager™, Compute Manager™, PBS™, PBSWorks™, PBS GridWorks®, PBS Professional®, PBS Analytics™, PBS Desktop™, PBS Portal™, PBS Application Services™, e-BioChem™, e-Compute™ and e-Render™ are trademarks of ALTAIR ENGINEERING INC.

Altair trademarks are protected under U.S. and international laws and treaties. Copyright© 1994-2014. Additionally, Altair software is protected under patent #6,859,792 and other patents pending. All other marks are the property of their respective owners.

ALTAIR ENGINEERING INC. Proprietary and Confidential. Contains Trade Secret Information. Not for use or disclosure outside of ALTAIR and its licensed clients. Information contained inHyperWorks® shall not be decompiled, disassembled, or “unlocked”, reverse translated, reverse engineered, or publicly displayed or publicly performed in any manner. Usage of the software is only as explicitly permitted in the end user software license agreement.

Copyright notice does not imply publication

Contents

HyperWorks 13.0 Release Notes	3
Software and Hardware 13.0 Release Notes	9
HyperWorks Licensing 13.0 Release Notes.....	12
HyperWorks Desktop 13.0 Release Notes	14
HyperMesh 13.0 Release Notes.....	15
HyperView 13.0 Release Notes.....	48
HyperGraph 13.0 Release Notes.....	66
MotionView 13.0 Release Notes.....	80
HyperCrash 13.0 Release Notes.....	104
HyperWorks Solvers	110
OptiStruct 13.0 Release Notes.....	111
RADIOSS 13.0 Release Notes	138
MotionSolve 13.0 Release Notes.....	149
AcuSolve 13.0 Release Notes.....	162
HyperXtrude Solver.....	174
BasicFEA 13.0 Release Notes	177
HyperWorks Collaboration Tools 13.0 Release Notes	180
HyperMath Solutions	182
HyperMath 13.0 Release Notes	183
HyperStudy 13.0 Release Notes.....	186
Manufacturing Solutions	189
HyperForm User Profile 13.0 Release Notes.....	190
HyperXtrude User Profile 13.0 Release Notes	194
HyperMold User Profile 13.0 Release Notes.....	200
Engineering Solutions	202
Aerospace 13.0 Release Notes.....	203
CFD 13.0 Release Notes	205
NVH 13.0 Release Notes	209
Crash 13.0 Release Notes.....	215
CAD and Solver Interfaces 13.0 Release Notes	218
Abaqus Interface 13.0 Release Notes	219
ANSYS Interface 13.0 Release Notes.....	225
CAD Interfaces 13.0 Release Notes.....	231

LS-DYNA Interface 13.0 Release Notes	233
Nastran Interface 13.0 Release Notes	242
PAM CRASH 2G Interface 13.0 Release Notes	249
RADIOSS Block Interface 13.0 Release Notes	251
OptiStruct Interface 13.0 Release Notes.....	255
Samcef Interface 13.0 Release Notes	270
Model Identification Tool 13.0 Release Notes.....	271
Virtual Wind Tunnel 13.0 Release Notes	278
SimLab 13.0 Release Notes	280
Evolve v2014 Release Notes	292
Inspire v2014 Release Notes.....	294

HyperWorks 13.0 Release Notes

HyperWorks is a comprehensive simulation platform for rapid design exploration and decision-making. HyperWorks provides a tightly integrated suite of best-in-class tools for all facets of the simulation process: modeling, analysis, optimization, visualization, reporting and collaborative simulation management. Leveraging a revolutionary pay-per-usage licensing model, HyperWorks delivers maximum value and flexibility for customers worldwide.

The 13.0 version of the HyperWorks suite offers many new functionalities in such areas as lightweight design, composites structures analysis, efficient modeling and meshing, multiphysics and multi-disciplinary analysis, structural optimization and design exploration.

Modeling and Visualization Capabilities

The HyperWorks products have gotten significant updates in usability. Across HyperMesh, HyperView and HyperGraph tools like the Entity Editor, Plot browser and the Advanced Build Plots panel ensure faster access to data and modification of multiple entities simultaneously. This release provides a variety of CFD related meshing feature updates and mesh quality review functionality. The already strong offerings for composites modeling have been refined with various updates and additions to existing features and procedures. Advancements in the tools for model assembly improve the handling of more and more complex models.

Usability

- The new **Entity Editor** provides an efficient way to create and edit cards in **HyperMesh**. Many attributes can be adjusted simultaneously across entities. Solver rules are captured to help set up new decks, and tooltips explain parameters without looking into the manual.
- Similarly **HyperView** offers an Entity Editor to modify attributes of multiple entities at the same time. **HyperGraph** introduced a Plot browser and an advanced Build Plots panel for efficient creation and manipulation of curves.
- Report generation just got easier with a completely new PowerPoint export functionality. A permanent link between **HyperWorks Desktop** applications and PowerPoint allows for updates of individual parts of the post-processing session.
- Markers can be tracked in **MediaView** to generate plots in **HyperGraph**.

Meshing

- **HyperMesh** offers two new meshing algorithms for CFD meshing. The Octree mesher is a faster and high quality alternative to the proven advancing front meshing algorithm and can be used for other than CFD applications as well. There is also a new boundary layers generator combining the smoothness of existing algorithms with added robustness and user control to handle any complex model.
- The Adaptive Wrapper mesher is a quick and robust way to generate a closed surface mesh for complex models while maintaining model details.
- The Quality Index panel offers a patch checker allowing you to cycle through areas with elements failing user defined criteria. Additional improvements for mesh modification helps remove these conflicts throughout the review process.
- The model can be contoured by user-defined quality criteria. The legend allows for detailed modification of display and criteria with innovative on-screen manipulators.
- **SimLab** received various CFD updates. A refined Mesh Wrapping algorithm for complete wrapping or connecting two parts enable the generation of closed outer surfaces. It is possible to extract the surface of enclosed volumes or outer surfaces for further meshing or to prove water tightness.
- Numerous additions to do modifications to parts of the mesh, such as adding ribs or removing multiple fillets, are provided with this release.

- For model checking purposes, a utility to find or identify duplicate or intersecting parts is available. **HyperMesh** and **SimLab** introduced comparison tools identifying differences between two meshes, CAD and meshes, or CAD amongst each other.

Composites/Aerospace

- **HyperMesh** now has a dedicated Aerospace user profile providing many tools and processes specific to the Aerospace industry, as well as composites applications of other verticals.
- The already efficient composites modeling process received a variety of useful updates. In addition to improved display and handling of plies and laminates, a review of orientations, elements, materials and ply systems is available in both HyperMesh and HyperView.
- **HyperMesh** includes an interface to Fibersim drape simulation data. With this release, results from composites modeler Simulayt can be read and seamlessly integrated into the composites workflow.
- **HyperMesh** features a new Drape Estimator to derive ply angle deviation and thickness variation due to draping.
- When composites analysis with shell elements isn't detailed enough, **HyperMesh** can convert a shell model into an equivalent solid model and automatically generate the necessary properties.
- **HyperView** offers functionality to generate Free Body Diagrams to analyze forces and moments in section cuts.
- Systems for bars can be plotted and results transformed between different systems.
- **HyperForm** supports multi-layered composite material forming for simple setups.

Assembly

- **HyperWorks** includes a material library that allows searching and filtering a corporate material database. The application can also be connected to the material database of our Altair Partner Alliance partner "Key to Metals".
- The ID Manager in **HyperMesh** organizes ID ranges for individual entities or include files and is available for the Nastran and LS-DYNA interfaces.
- A new and comprehensive part replacement tool and find functionality for tied contact definitions complement the LS-DYNA interface.
- The Module entity includes numerous updates to organize assemblies, define a current modules and to visualize content in the Model browser. Cross-referencing is also enabled for modules.

Analysis and Optimization

HyperWorks solver technology includes finite-element-based linear and non-linear structural analysis, design and optimization capabilities (OptiStruct), finite-element-based highly nonlinear structural analysis under dynamic loading (RADIOSS), thermal and fluid analysis (AcuSolve) as well as multibody simulation (MotionSolve). Combined with design optimization technology and multi-physics capabilities, HyperWorks enables users to drive the product-development process and make reliable decisions based on high-quality results.

With the HyperWorks 13.0 release, Altair continues a successful mix by adding new functionality to accommodate customer needs and latest research results, combined with usability enhancements and performance improvements by leveraging the latest computing technology.

Optimization

Many new capabilities have been introduced to reinforce Altair's leadership in simulation-driven design and design-oriented, optimization-enabled solutions:

- **OptiStruct's** multi-model optimization allows running multiple models with common design variables simultaneously
- The use of Excel spreadsheets for external response definition in **OptiStruct** is enabled

- A new, reliability-based optimization technique is available as BETA capability in **OptiStruct**
- Optimization problem setup and solution is possible in the native **RADIOSS** input format

Multiphysics

An expanded number of solution types and material models broaden HyperWorks' multi-physics offering:

- Native large displacement (quasi-)static analysis with hyper-elastic and elasto-plastic materials as well as small sliding surface-to-surface contact is available in **OptiStruct**.
- **OptiStruct** also offers nonlinear heat transfer analysis, as well as a new rotor dynamics, for frequency response and complex eigenvalue analysis.
- In **RADIOSS**, enhanced airbag modeling regarding finite volume gas modeling, contact, material and element formulation is now offered. New material laws for foams, polymers, and hot stamping and enhanced failure modeling thru XFEM are available.
- Direct coupling of multibody dynamics and CFD is possible via **MotionSolve** and **AcuSolve**.
- **MotionSolve** adds a new general multi-purpose bushing model and an improved workflow for co-simulation with Simulink.

Performance

Altair HyperWorks utilizes parallel computer architectures for enhanced solvers performance:

- In OptiStruct, parallelization using Domain Decomposition for scalability and speed of linear and non-linear static solutions and optimization is available.
- Parallelization for multi-domain method has been improved in **RADIOSS**. Further performance improvements have been achieved through Advanced Mass Scaling.

Usability

HyperWorks 13.0 introduces many new and enhanced capabilities to streamline analysis model setup:

- In **OptiStruct** models can be created in terms of parts and instances, along with global/local analysis for enhanced accuracy of local results.
- A brand new contact definition (Type 24) for zero gap solid-to-solid contact modeling has been introduced in **RADIOSS**.
- AcuSolve provides interpolated mesh motion.
- Overall usability has been improved via detailed error messaging, model checking, result output, documentation, and tutorials.

Value-based Business Model

Hosted HyperWorks Units (HHWU)

- Altair innovates its HyperWorks licensing model by hosting licenses on the cloud, called Hosted HyperWorks Units (HHWU). This offering provides users with instant portability to their HyperWorks units, eliminating the IT burden of maintaining a license server in-house.
- Continuing the legacy, HyperWorks 13.0 expands the availability of HHWU to the next-generation, cloud-based licensing system. In this new system, HyperWorks can be accessed from anywhere.
- HHWU allows increased portability and scalability and reduced IT burden in maintaining a license server.

- Please contact an Altair representative to learn how to take advantage of the added value and how your current licenses can be moved to the cloud.

Cloud Solutions

- Altair's CAE cloud solutions capitalize on our unique position in the industry and experience in offering engineers and scientists the flexibility to run simulations. Users are able to run CAE simulations instantly on their desktops anywhere by leveraging our Hosted HyperWorks Units (HHWU). Customers can further extend their experience with enterprise capabilities to scale on-premise (HyperWorks Unlimited) or scale out to the cloud (HyperWorks On-Demand), while simultaneously keeping track of the entire simulation data with Altair's Simulation Cloud Suite. In all of the cases, users are provided with a comprehensive and intuitive web-based interface for managing data and knowledge in the cloud.
- Altair's technical cloud solutions address the unique needs of enterprises by simplifying access to an HPC infrastructure, allowing engineers and scientists to focus on their work rather than deciding how and where their jobs are executed.

Partner Alliance

The Altair Partner Alliance continues to expand its portfolio of third-party technology.

- Applications from upwards of 40 strategic partners can be directly accessed with HyperWorks Units in the following domains: durability and fatigue, computational fluid dynamics, impact analysis, noise-vibration-harshness, electromagnetic analysis, 1D systems simulation, injection molding simulation, process structure interaction, composite modeling, stress analysis, material libraries, thermal analysis, complexity management, casting simulation, multi-body dynamics analysis, project management, rendering, and rapid prototyping.

Ideation and Conceptual Design

solidThinking Inspire

solidThinking Inspire enables design engineers, product designers, and architects to create and investigate structurally efficient concepts quickly and easily. Inspire enhances the design process by generating a new material layout within a package space using the loads as an input. The software is easy to learn and works with existing CAD tools to help design structural parts right the first time, reducing costs, development time, material consumption, and product weight. Major new capabilities in solidThinking Inspire 2014 include:

- Geometry Simplification – Clean up and defeature problem areas in the geometry including imprints, rounds, fillets, holes, and pocket.
- Analysis – Investigate linear static and normal mode analysis on a model
- Displacement constraints – Displacement constraints can be applied to a model to limit deflections in desired locations and directions.
- Smoothing options – Create surface models from Inspire and export as a geometry file.
- Concentrated Mass Part – Connect concentrated mass either on a part or at a point in space.

solidThinking Evolve

solidThinking Evolve allows industrial designers to develop forms faster, using either Windows or OS X. It enables you to capture an initial sketch, explore styling alternatives, and visualize products with realistic renderings generated in real time. Evolve provides organic surface modeling and parametric control, with NURBS-based surfaces and solids and a unique ConstructionTree history feature. Major new features in solidThinking Evolve 2014 include:

- Improved modeling tools – Refined replication tools for a more intuitive workflow, increased surface control for the Round tool, and more robust deformation tools.
- Live linking to KeyShot – Enables KeyShot scenes to be continuously updated directly from Evolve.

- New construction aids and scene management – New and improved snaps, new isolate mode, better management of hiding/showing objects, and many new hotkeys to increase workflow efficiency.
- Enhanced user interface options – Simplified workflow for importing and controlling background images, new options to enable/disable selection pre-highlighting, and a mode to quickly disable curve visualization for better model evaluation.

Design Exploration

Expand your horizon with HyperWorks design exploration tools; performing “what-if” studies, understanding and optimizing system behavior with HyperStudy and performing complex custom calculations with HyperMath.

HyperStudy continues its fast pace of development based on the updated product released in version 12.0. Some of the new functionality HyperStudy 13.0 offers are:

- Simplified study setup via enhanced model definitions and a new, powerful Parameter Editor
- Trade-off studies portable to Excel spreadsheets
- Localization of user interface. Languages added to HyperStudy are: Japanese, Korean, Chinese, German and French
- New ANSYS Workbench model type
- Simplified optimization setup simplified via automatic method selection and reduced display of method settings

Engineering and Manufacturing Solutions

HyperWorks 13.0 extends the offer for manufacturing and engineering solutions, with new functionalities and an innovative delivery model to better capture knowledge for vertical processes.

HyperCrash

- **HyperCrash** is a dedicated tool for typical crash analysis and safety evaluation. In this release, support for RADIOSS 13.0 has been added. For dummy seating, pre-simulation can be used to adjust the seat to the dummy shape or to avoid intersections of the dummy while positioning. HyperCrash offers also support of parameters for RADIOSS and LS-DYNA solvers. Conversion of LS-DYNA to RADIOSS models has been enhanced.

HyperWorks Manufacturing Solutions, a unified environment for manufacturing process simulation, analysis, and design optimization, includes enhanced solutions for sheet-metal forming, metal and polymer extrusion, friction-stir welding, injection molding and forging.

HyperForm

- Altair HyperForm is a comprehensive FE-based sheet metal forming simulation framework. In version 13.0 an intuitive interface for setting up complex multi-stage stamping processes has been added. The coupling of HyperForm with the design optimization tool HyperStudy and our explicit solver RADIOSS makes stamping process optimization possible. HyperForm now also supports hot stamping and a simple setup for composite material stamping. The mass scaling with adaptivity has been significantly improved.

The HyperWorks Solutions are specialized packages of software and services that support the implementation and utilization of Altair’s solutions and technologies, tailorable to meet customers’ specific requirements and automate CAE workflows. Solutions include:

Reporting Automation Director

- Automated Reporting Director streamlines the often time-consuming tasks associated with the mechanics of results generation and reporting, allowing focus to remain on the interpretation and understanding of results.

NVH Director

- NVH Director is designed to reduce the simulation cycle time, freeing CAE engineers to focus on optimizing product design and performance.

Impact Simulation Director

- Impact Simulation Director solutions automate the laborious, manual tasks associated with model set-up, analysis, post-processing and reporting.

Squeak & Rattle Director

- Squeak and Rattle Director offers a complete set of capabilities to streamline the entire S&R simulation workflow process from model creation to results visualization.

More solutions are currently under development. Please contact an Altair representative to learn more about the available HyperWorks Solutions and how to use them with your current licenses.

For HyperWorks framework level release notes addressing topics such as hardware support and application licensing, see [Software and Hardware Release Notes](#) and [Altair Licensing Release Notes](#).

Software and Hardware 13.0 Release Notes

Introduction

Various changes have been added to HyperWorks 13.0, including the addition of new products, hardware support, invoking the package and installation.

Hardware Platforms

Operating System Additions

Support for the following platforms has been added:

- Windows 8.1 64-bit (x86_64)
- Red Hat Enterprise Linux 5.9 64-bit (x86_64)
- Mac OS X 10.8 64-bit (x86_64)

Discontinued Hardware Platform Support Systems

Support for the following platforms has been dropped across all HyperWorks applications:

- Windows XP (both 32-bit and 64-bit)
- All 32-bit architecture support
- Red Hat Enterprise Linux 5.8 64-bit and lower
- Mac OS X 10.7 64-bit (x86_64)

Installation

HyperWorks 13.0 suite contains several independent product packages. This gives you the flexibility to download desired packages as needed. Any HyperWorks 13.0 standalone SA update package can be installed either on top of an existing 13.0 installation, or can be installed independently without the prerequisite of a 13.0 base. Multiple product SA updates can now be installed concurrently with or without the 13.0 base.

HyperWorks 13.0 installer brings the robust capabilities and features of the HyperWorks 12.0.1 installer as well as the additional support for the HotFix update installation. The 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 include the quick fixes and features to better suit your needs.

Installation and Packaging

- Added Altair digital signature to the Windows HyperWorks packages
- Added support for HotFix (quick patch) update installation wizard
- Added the SimLab product on Windows and Linux (included in HyperWorks 12.0 SA)
- Added Virtual Wind Tunnel for Windows (included in HyperWorks 12.0 SA)

Installation/uninstallation

- Upgraded the HyperWorks installers to the InstallAnywhere 2012 SP1 web installers
- Added Windows 8 and Windows Server 2012 support. HyperWorks only officially supports Windows 8.1
- Improved the Linux console mode installation
- Improved the uninstaller icon warning. Some Windows systems were missing a specific version of the WinSxS MSVC++
- Better handling of SMPD and MPI service during an AcuSolve installation on Windows

Invoking

- Added a set of menu items for Linux operating systems. These are installed inside the Altair home folder. When the menu installation script is run, they are placed into the Linux System menu allowing users to launch HyperWorks applications similarly to the Start Menu on Windows. These menu items work with both KDE and Gnome and conform to the XDG desktop standard.
- Added better version information to the shortcuts for the Windows 8.1 Start Menu and Metro Tiles
- Added better font size management for the HyperWorks Desktops applications, including HyperMesh under Linux and Mac OS X user environment where the default size is set to a 9 point font

Resolved Issues

- Fixed issues on Windows and Linux only where the Summary panel available disk space usage reported a "-0" on very large hard disks
- On Linux, when installing using the console mode, the Help Configuration prompt will no longer add another set of double quotes if the **Back** command is used.
- On Linux, there are no longer duplicate installation bars when the **Back** command is used from the Summary panel
- Correct Linux and Mac OS X "read + execute" permissions of the sub installers when using the master installer

Known Limitations in HyperWorks Installer 13.0

- In Mac OS X 10.8 and higher the installer will still report "-0" on the Summary panel due to a bug in InstallAnywhere 2012 SP1.
- Using the **Back** option during an installer may not reset the installer variables correctly.
- On Linux and Mac OS X, installing in a directory with spaces is not supported.
- The installer will not allow users to go back past the Install Pick List panel.
- On Linux, during a console mode install, when using a response file output the Help Configuration in the file may not write out correctly.
- If there is not enough TMP/TEMP space on the main drive (that is, C: under Windows), then the installer will prompt for the user to input to a new location. Some Java TMP/TEMP space will still try to go to the main drive even when selecting this new location.
- When clicking on **Cancel** to stop an installation via the Master installer, the files that were already installed are not cleaned up, but the files in the TMP/TEMP folder *are* cleaned up. You will need to uninstall/delete the installed files from the chosen install folder manually if **Cancel** was used.
- The HyperWorks installers do not support the use of the default InstallAnywhere `installer.properties` files. We request that a unique name is used when using a response file,

(for example, `installer_hw12.properties`) and to use it with the `'-f <path_to_file>/<filename>'` command line option.

- For NETSETUP client installs, a UNC path must be used to get to the server installation path through Windows Explorer or Windows Network. Note, using mapped drive letters or variables is not currently supported in the installer.
- With some versions of Gnome, there is a bug where the icons are not placed in a hierarchy. Instead, they are placed in "Other" with a flat layout, meaning that all the icons appear at the same folder level in "Other."
- The installer does not run the icon installation by default; the user must run the icon installer.

HyperWorks Licensing 13.0 Release Notes

Introduction

The HyperWorks 13.0 suite of products is licensed using the Altair Licensing System which provides a unique and flexible Altair HyperWorks licensing model to our customers. This gives full flexibility and access to all HyperWorks products and to partner products through the HyperWorks Partner Alliance (<http://www.hyperworksalliance.com>). The HyperWorks licensing model supports large scale computing, design studies, multiphysics, and optimization by decaying the license draw with increasing usage.

Application Unit Draw

Changes made to HyperWorks 13.0 product license unit draw:

- SimLab version 13.0 is included. 25 HWUs are required to run SimLab Meshing, Assembly and Results applications. 40 HWUs are required to run SimLab Automation and DOE.
- Virtual Wind Tunnel 13.0 is included and requires 21 HWUs to run this application.
- TableView, a new HyperWorks desktop client, is included and requires 6 HWUs.
- HXProcess, a new solver for HyperXtrude, is included and requires 10 HWUs. See the Solver Unit Draw table below.

Solver Unit Draw

Below are changes to license features used in HyperWorks Solvers 13.0.

HyperWorks Units	Feature Name	Functionality
1	OptiStructCheck	OptiStruct Check run
	RadiossCheck	RADIOSS Starter
10	HXProcess	HyperXtrude Process
25	Radioss	RADIOSS Engine
	OptiStructFEA	OptiStruct (Analysis only)
	AcuSolve	AcuSolve
	AcuView	AcuView
	AcuTrace	AcuTrace
	AcuFwh	AcuFwh
	MotionSolve	MotionSolve
50	OptiStruct	OptiStruct Optimization

	HyperXtrudeFEA	HyperXtrude Solver
--	----------------	--------------------

Compatibility

HyperWorks 11.0, 12.0, and 13.0 are compatible with Altair License Manager 12.0 (and updates). Using the latest update with HyperWorks 13.0 is recommended.

Resolved Issues

- Fixed memory leak in Hosted HyperWorks Units (HHWU) checkouts
- Increased stability in Tri-server (HAL server) setup
- Fixed issue with non-writable temp directories

HyperWorks Desktop 13.0 Release Notes

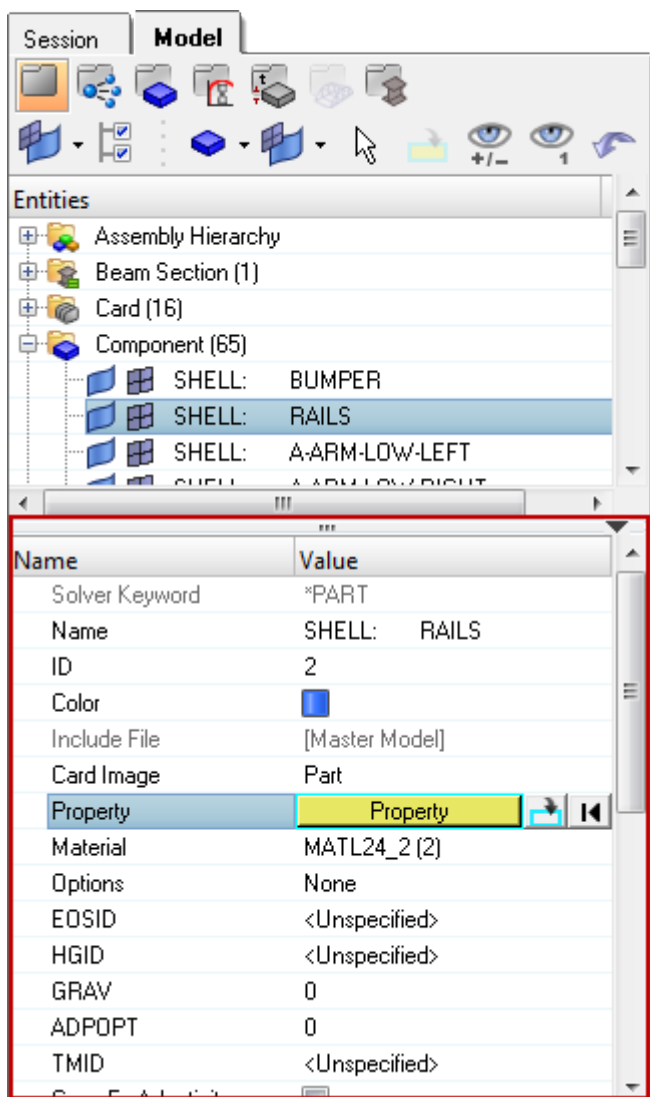
HyperWorks Desktop is an integrated user environment for modeling and visualization. It can be used to pre-and post-process finite element and multibody dynamics simulations as well as to manage and visualize test data. The following applications are part of the HyperWorks desktop framework: HyperMesh, MotionView, HyperView, HyperGraph (2D and 3D), MediaView, and TextView. HyperWorks Collaboration tools are fully integrated in this programmable and open environment. HyperMath is a scripting language and a fully integrated IDE for scripting and debugging using HML or TCL. HyperMesh is available in the Desktop as well as in the traditional standalone mode.

HyperMesh 13.0 Release Notes

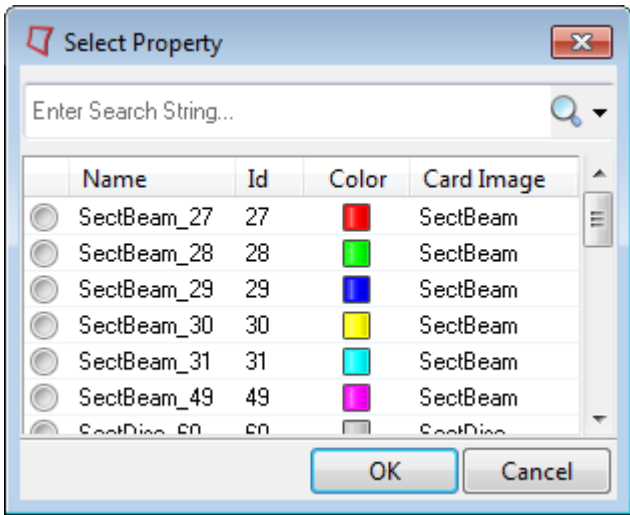
General

Entity Editor

The Entity Editor is a new usability update to enable rapid creation, editing and query capabilities. The Entity Editor can be found within a large majority of browsers across HyperWorks 13.0; more specifically, HyperMesh 13.0 Model browser, Solver browser and cross-referencing tools are Entity Editor enabled. The following user profiles are supported: AcuSolve, OptiStruct, RADIOSS, Abaqus, ANSYS, LS-DYNA and Nastran. The Entity Editor is designed to provide instant feedback based on the selected entity in the browser and is complementary to the card edit functionality.



Additional usability updates within the Entity Editor include a new entity list selection mechanism which enables full sort, find and filter. Working with large data sets is now significantly more efficient.

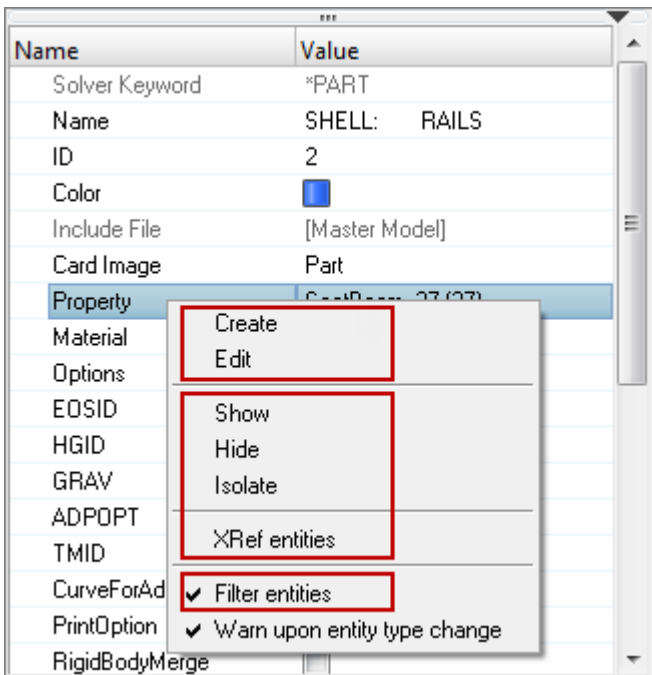


More support for browser entity creation is available – more entities across more user profiles can be generated via the context menu in the browser. In addition, work flow is improved, the number of mouse clicks has been reduced by at least half when creating new entities when compared to the traditional panels.

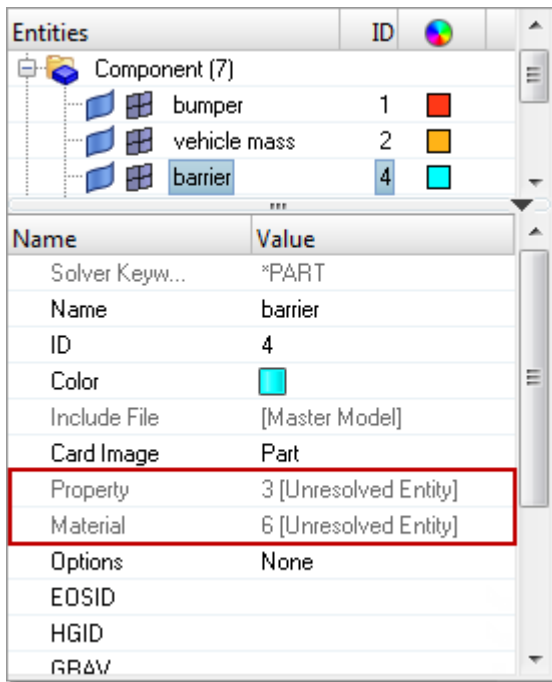
The ability to create, edit and cross-reference a referenced entity in the Entity Editor is enabled. As an example, a component may be selected and within this context you can edit the material and/or property directly via the Entity Editor context sensitive menu.

Additional filtering is enabled by default preventing an incorrect assignment of one entity to another.

Display control of referenced entities within the Entity Editor is also available.



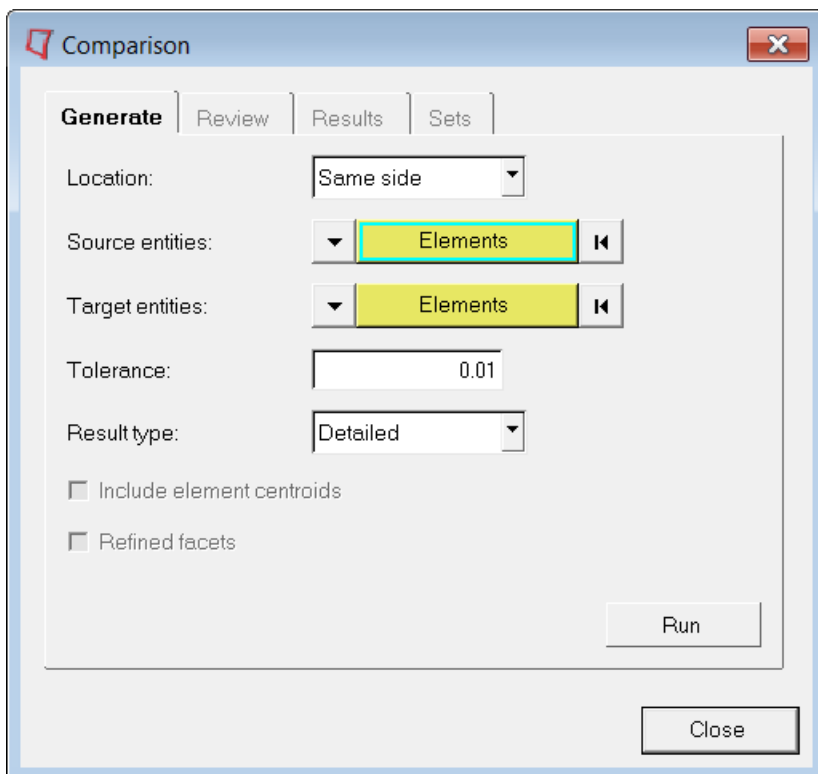
Unresolved ID's are now shown clearly in the Entity Editor, further improving the feedback mechanism to you.



Comparison Tool

The Comparison tool performs a model-based CAD-CAD, CAD-FE or FE-FE comparison between two models, or two selections of entities. You can also use this tool to find and report on geometrical/shape differences. To access the Comparison tool click **Geometry > Check > Surfaces > Comparison** or click **Mesh > Check > Elements > Comparison**.

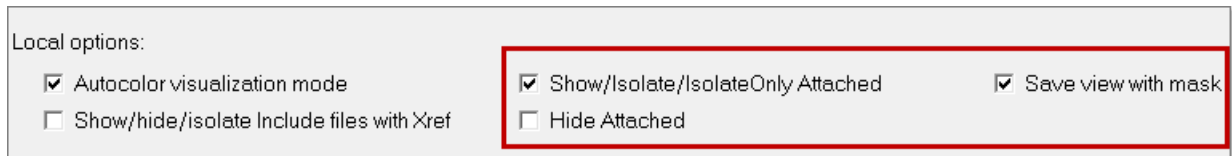
When you are comparing entities, the entities must exist in the HyperMesh database. The Comparison tool generates results by comparing the source entities to the target entities. Currently, only 2D elements are supported for the Comparison tool. The results generated by this tool can be broken down into the following categories: Matches, Overlapped, Intersected, and Unmatched.



Browser Display Controls

There are three new options added to control the display of entities when performing show, hide, isolate and isolate only operations. These new controls can be found by right-clicking in the Model browser and selecting **Configure browser**.

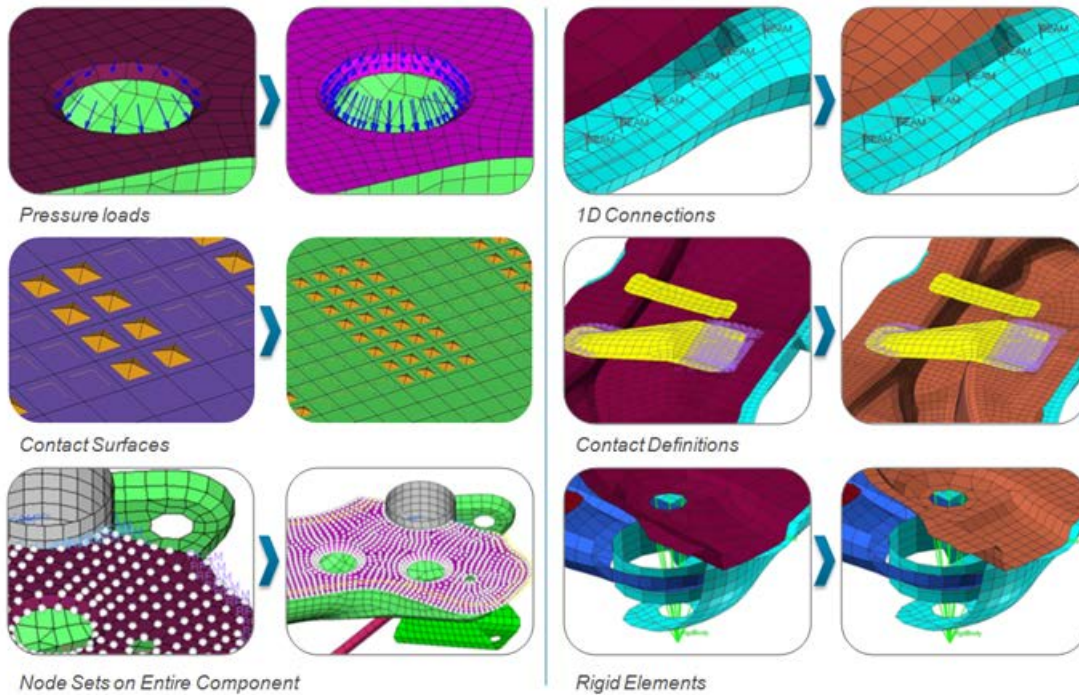
- Show/Isolate/IsolateOnly Attached - will show also attached entities. As an example, if a contact is isolated, the respective elements will also be shown.
- Show/Hide/Isolate include files with XRef – will perform the display operation of the include and will also display or hide the cross referenced entities related to the selected include(s).
- Hide Attached – will hide the attached entities. As an example, if a contact is hidden, the respective elements of that contact will be turned off from the display.



Part Replacement

The LS-DYNA user profile has a new and extensive part replacement tool. The tool handles all referenced entities when migrating from the old to the new part. There are a number of options to decide whether to replace a part within the session, or, point to an external file. A preview and check capability are added to ensure successful part replacement before the actual operation. Many entities and organizational constructs are supported during replacement including:

- Maintain property and material ID
- Loads and boundary conditions
- Sets
- Contact surfaces
- Include file organization
- Output blocks
- Control volumes
- Systems and vectors
- Connectors, rigid elements and masses



Preserved Nodes Panel

When a free node is not referenced by any of the entities in your model, HyperMesh automatically deletes the unused nodes. You can save a free node that you are not currently using by converting it into a temporary node, but HyperMesh will only continue to save these free nodes as long as they are marked as a temporary node. To guarantee that a free or temporary node will be saved in your model once you create them, you must convert them into a preserved node inside the Preserved Nodes panel.

Favorite Panels

A new "Favorites" toolbar and functionality for saving favorite panels is added.

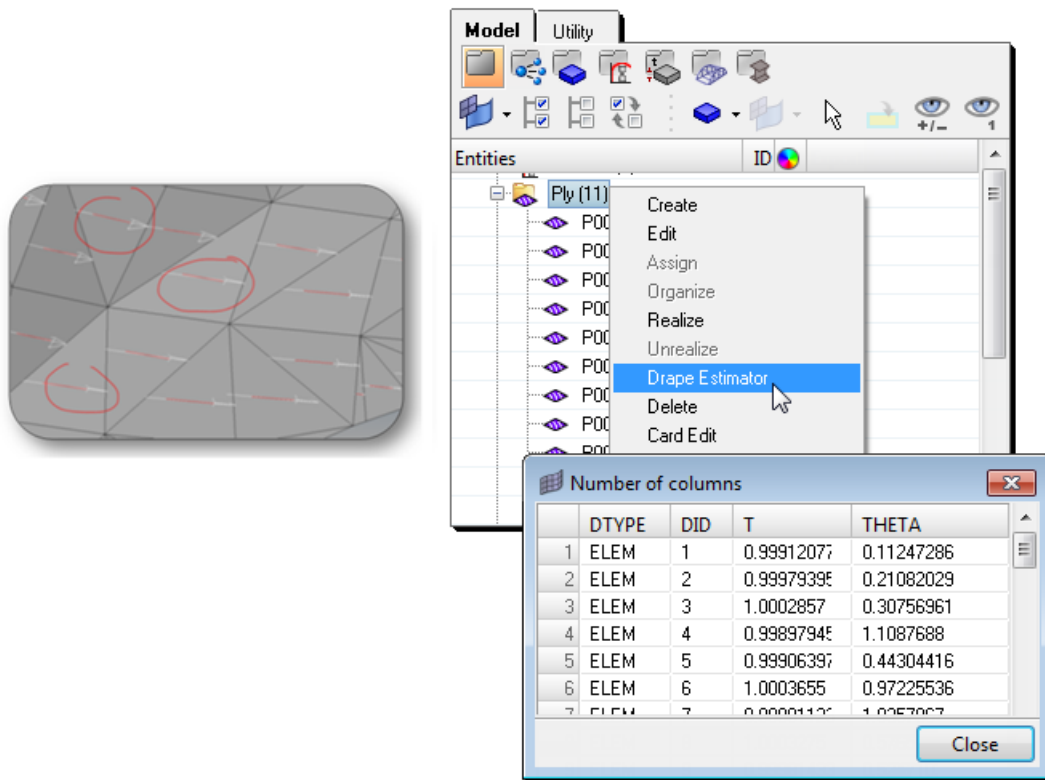


The Favorites panel allows you to:

- Add and remove your favorite panels for easy access
- Save your favorite panel list to hmsettings.tcl

Drape Estimator

The Drape Estimator is a tool to generate directly in HyperMesh (OptiStruct and Nastran user profiles) draping angles and thickness variation resulting from the manufacturing process for fibers associated with plies in a composite laminate. You can use the drape data generated by this tool based on inverse mapping to improve the accuracy of parts modeled with composite materials.



Miscellaneous Updates

- Titles are now supported also for surfs
- H3D export has been enhanced:
 - H3D export now honors the entity visualization and transparency state
 - The resulting H3D file can also now contain both elements and geometry (STL) in the same file
 - Titles associated with exported entities are now written to the H3D file

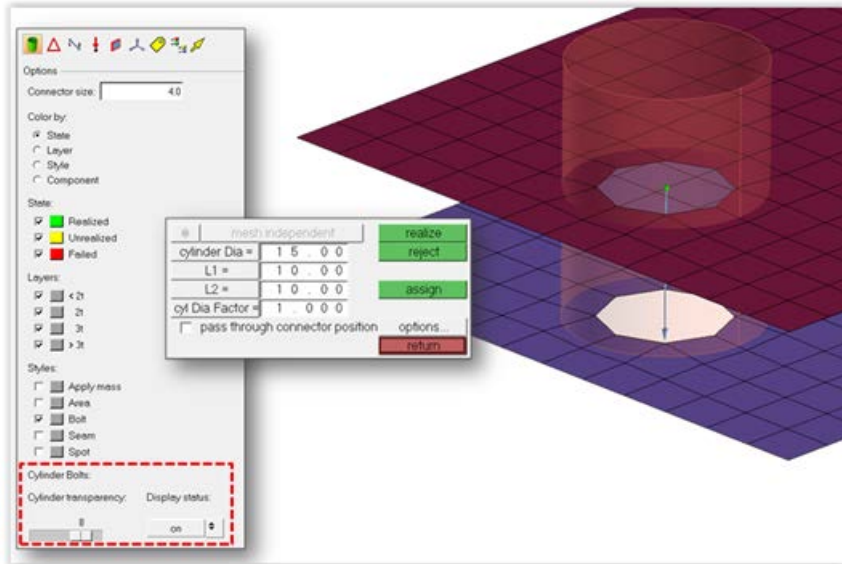
Resolved Issues

- Box trim:
 - Removed “Box Collector” field from Box Trim GUI for LS-DYNA, Nastran and OptiStruct user profiles
 - Removed temporary nodes which are outside the bounding box for LS-DYNA and Nastran user profiles
 - Box Trim dialog closes after trim operation for LS-DYNA and Nastran user profiles
 - A cleaner and more error free model is produced as empty collectors, empty contacts and other unused entities are deleted at the end of the box trim process.
- Show, Hide, Isolate control on includes no longer shows the associated plot

Connectors

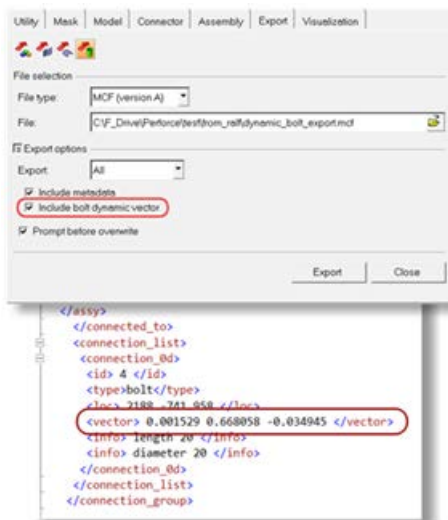
New Bolt Preview

The new bolt preview enables a cylindrical preview which highlights which nodes will be included within the bolt realization. There is also a vector preview. Both can be managed as a temporary or permanent mode with additional transparency control.



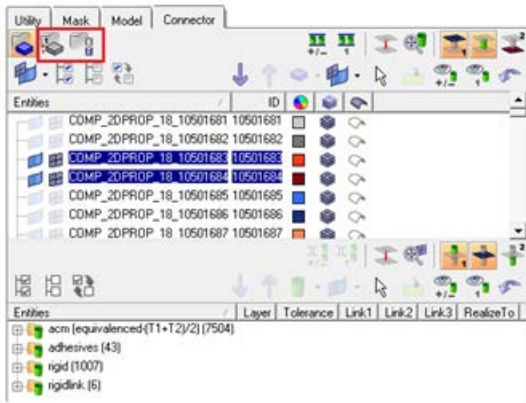
Export Dynamic Bolt Vector

Connectors will be exported with the dynamically identified bolt connector vector by default.



Property Link View and Modules Link View

Property link View and Modules link view are two new predefined browser view modes now available in link browser view section of Connectors browser.

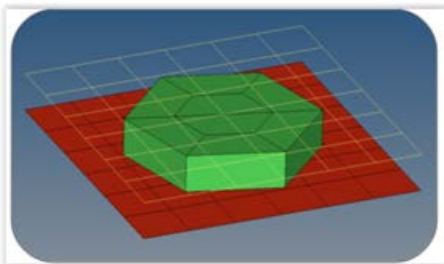


Updates to the FE Config File

- Fe config files will be searched/loaded in the sequence of installation folder:
 - The folders defined by HW_CONFIG_PATH environment variable
 - HOME directory
 - Working directory
- Instead of just displaying the FE types from last loaded FE configs file, now all FE types will be available through various loaded FE config files. During the load of various FE config files, the conflicting FE types would be skipped.
- Search sequence for the Property scripts associated with a FE type available in a FE config file would be in the order of working folder, user HOME folder, folders defined by HW_CONFIG_PATH environment variable and installation folder.
- A FE config radio button is added to the Connector Options subpanel, which lets you load or unload a FE config file during the session.

New Realization Support and Updates

- RBE3 spider/wagonwheel bolt realization.
- CGAP(G) realization for OptiStruct user profile
- Hexa Spot with contact and material (law59) realization for RADIOSS user profile

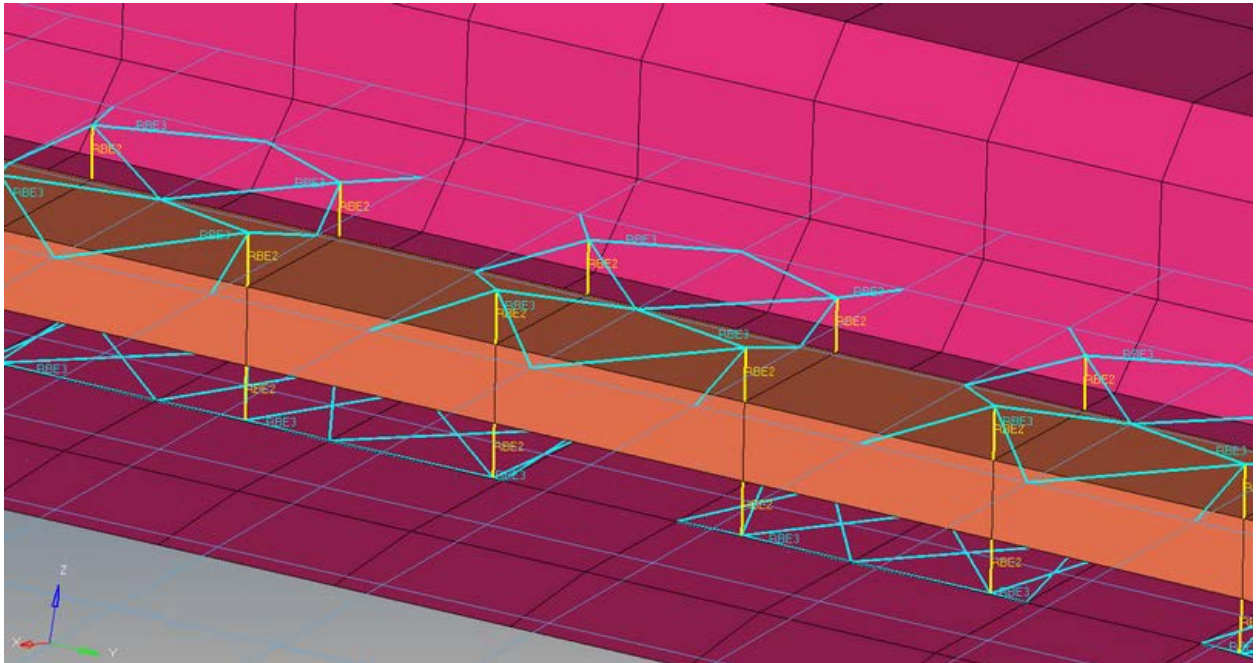


Connector Location Information Available as Metadata

The connector location information can now be retrieved from the individual connector as a metadata value. For example, if a connector is created with the nodes as location, those nodes IDs can be retrieved from the connector.

Resolved Issues

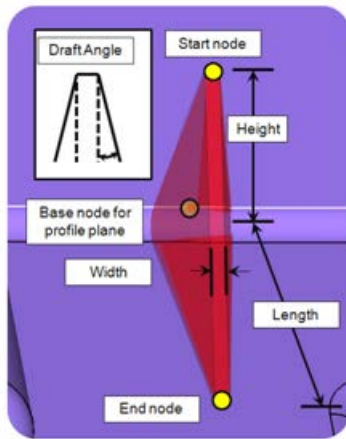
- Seam hexas made perpendicular to the seam line.
- CWELD card details for "GS ELEMID" realization with node as link rectified.
- Connectors lines are missing internal biasing during export and later import of the model.
- CBAR realization was assigning 123456 dof to PIN A in the card image.
- Seam with imprint was not capturing the spacing correctly.
- Changing Element filter options in FE Absorb adhesives was giving an application error.
- Some issues related to direction and location of ACM realization.
- T connection adhesive realization with constant thickness earlier with bad quality welds is rectified now.
- The FE register api "`*CE_FE_RegisterAdvanced`" now registers FE's at a specified test point locations.
- Hillock CBUSH stiffness values rectified in some cases.
- Realized element will be excluded from the projection.
- Add link to area connector [to existing] is rectified.
- Absorption and Re-realization of 2-layer single link connection for mat100(HEXA)/rigidlink config for LS-DYNA was not working and has been rectified.
- Absorption of Solid-Seams configs ie with RBE3-RBE2-hexa-RBE2-RBE3 was not working and has been rectified.



Geometry

New Geometric Rib Tool

A new geometric rib tool is now available that enables creation and modification of simple ribs between surfaces. Several rib parameters such as height, length, and width can be controlled interactively. The newly created rib is automatically stitched into the adjoining solid and/or surfaces. Both triangular rib for T-joints and quadrilateral ribs for U-sections are supported.



Midsurface Generation Tools

New tools have been added to generate midsurfaces using pieces of "sweep surfaces" to build a midsurface whenever it is possible and appropriate. A sweep surface is a surface obtained by moving a straight line along an arbitrary planar curve. A cylinder is a good example, where a circle is dragged along a line perpendicular to the circle plane to create the cylinder.

This is in addition to the previously released Insert Planes functionality (12.0) which allows using pieces of planes, and the legacy Offset method.

The default option for midsurface extraction is now "offset+planes+sweeps". This allows different regions of a single model to use different methods (offset, insert planes, insert sweeps) as possible and appropriate, with the results of each region connected together.

Midsurface Plate Editor

Enhanced the Midsurface Plate Editor to allow you to update the middle surface according to a user-defined plate edit in one click. If you have done some plate editing, clicking Update does the following:

- Deletes the pre-existing middle surface (if any)
- Deletes the display components showing the current plate information
- Re-extracts the middle surface in accordance with the user-defined plate edit and provided midsurface extraction setting
- Re-draws the updated plate information components

Metadata to CAE Tool

A new Metadata to CAE tool has been created to assist in utilizing metadata coming from CAD import.

Supported operations include:

- Renaming and renumbering components using associated metadata
- Creating spot, seam and area connectors by selecting points, lines or surfaces using their associated metadata

CAD Reader Updates

Newly supported formats:

- CATIA Composite Link drape data
- CATIA v5-6R2013 (v5 R23)
- Creo 2.0 M060

- JT 10.1
 - Parasolid v26 (native reader)
 - SolidWorks 2013
 - UG NX 9.0 (native reader) and NX 8.5 (third-party reader)
 - UG NX 9.0 now requires the license server to be defined using the SPLM_LICENSE_SERVER environment variable, while older NX versions still require the UGS_LICENSE_SERVER environment variable
-

CAD Writer Updates

- JT export of geometry and FE data
-

Resolved Issues

- Performance issue related to CATIA CPD file import
- Import issues with CATIA V5R21 and R19 files
- IGES export issue related to layer and groups selection was not correctly associated to the component ID
- Application crashes while using “Import Hidden Entities” option for a CATIA file on MAC machine only
- Metadata information was not importing if the same file is loaded for second time
- Distance option for trim with lines selection in surface edit panel was not working correctly
- Native Parasolid reader import option was missing for Tetramesh Process Manager and BatchMesher
- Missing plies with FiberSim import
- Application crashes while using offset subpanel of Surface Edit panel with some specific models
- With specific models, the JT reader crashes during the import
- Surface normal issue with JT import with specific models
- Application crashes while using solid trimming with some specific models
- Parasolid Solids with internal surfaces were not imported correctly
- Some crashes related to Autocleanup panel in bead handling and fillet trimming
- Error Importing JT assembly with specific models when scale value of greater than 1 is used.

HyperBeam

New Options

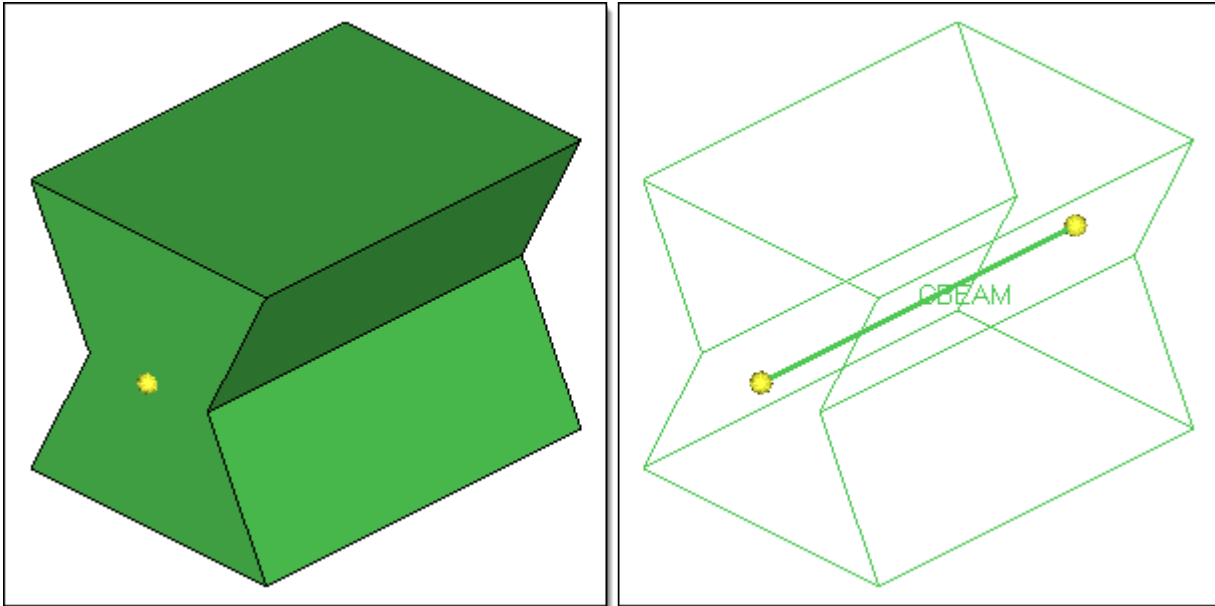
- **Delete all Sections** and **Delete all Sections and Collectors** is added under the File Menu.
 - **Copy** and **Paste** is added to the right-click context sensitive menu on a beam section in the Sections browser.
 - **Section Cut** is added for the option Shell Section at 1D > HyperBeam panel. Section cut option takes uses elements, surfaces or solids as input.
-

Shell Beam Section Creation

When creating a shell beam section for **Use lines** or **Use elems option** the thickness is extracted from the shell section instead of being set to 0.1.

3D Visualization of Generic Beam

New 3D visualization is added for generic beam sections. Generic beams will now appear with an hourglass shape provided that they have either the Centroidal, Principal, or Local moments defined, or they have the area defined. Otherwise they will display as lines with a coordinate system. The size of the beam will roughly approximate a rectangular section with the given moments.



Resolved Issues

- Naming for shell section parts is off
- Updates for the Moments tool interface

HyperMorph

Resolved Issues

- The reflective symmetries (1-plane, 2-plane, 3-plane, and cyclical) will now work for map to geometry and radius change provided that the morphed domains have the symmetry assigned to them.
- There is a minor change to the Symmetry panel. The checkbox now reads "morph volumes & mapping", meaning that, if checked, the symmetry will be applied to mapping operations performed without domains.

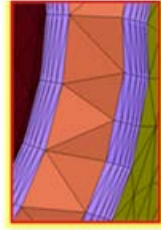
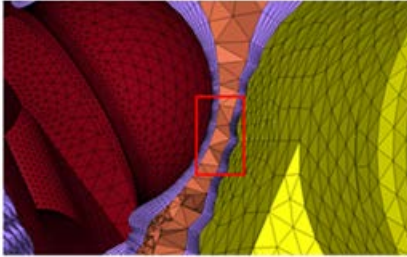
Meshing

New Meshing Method: Layered Tetra

A new method is implemented to generate multiple user-specified tetrahedral layers in thin regions of solids.

Min tetra size:
 Number of layers:
 fill voids fix midnodes smoothing
 Elem quality target:

 Refinement box:



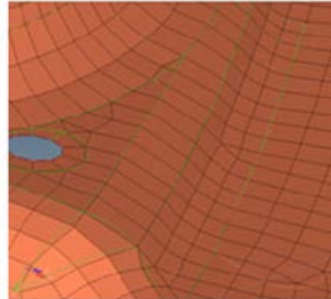
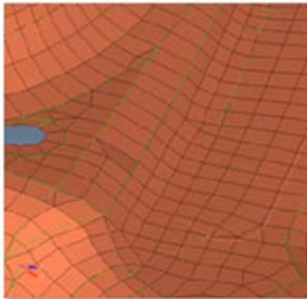
Layered Tetras Disabled



Layered Tetras Enabled

BatchMeshing

- Improvements focused on geometry feature recognition, which aim to better prepare the geometry for meshing. The new option **recognize and preserve major feature edges** in the BatchMesh parameter file enables this functionality, and includes:
 - Recognition of major, flat surfaces and preservation of their boundaries for both geometry and element cleanup. This should result in less “dissolving” of features.
 - Improved recognition of complex fillets and fillet chains.
 - Improvement and fixes for element cleanup to handle narrow areas between features.

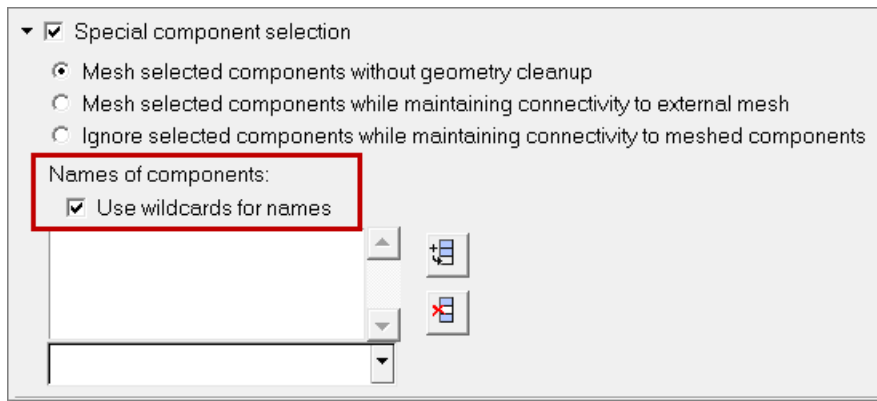


Preserve boundaries between components
 Recognize and preserve major feature edges

- Addition of a new option “remove trias attached to holes” that eliminates tria element edges from holes.

Create mesh with element type:
 Element order:
 Place elements in:
 Apply tria reduction with min elem size >
 Remove trias attached to holes
 Apply optimized smoothing:

- Wildcard for special component selection

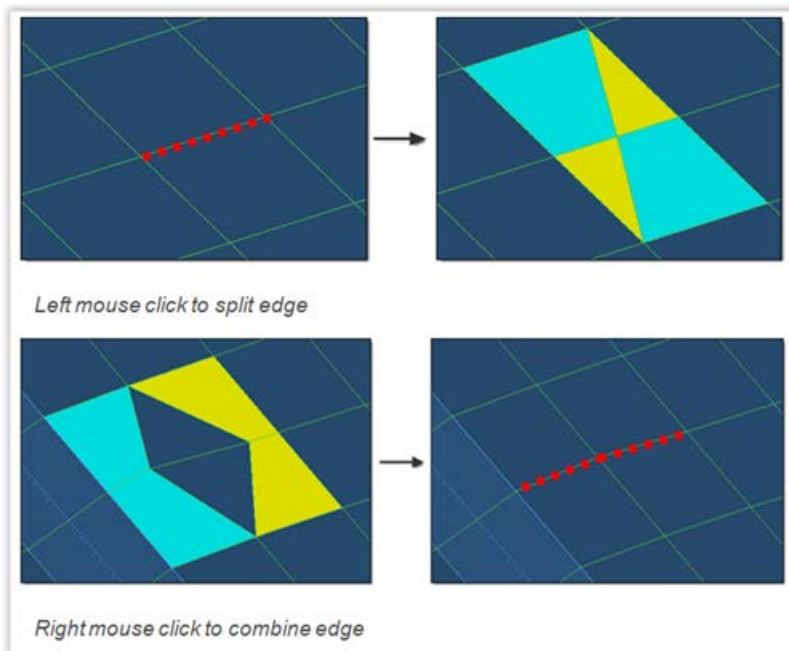


Quality Index Panel Enhancements

A new patch checking capability is available which provides a more efficient way of cleaning up undesirable mesh quality. The patch checker isolates the failed elements, you can traverse from one failed patch of elements to the next, and clean up the mesh by using one of the many cleanup tools from within the Quality Index panel.

New cleanup tools include:

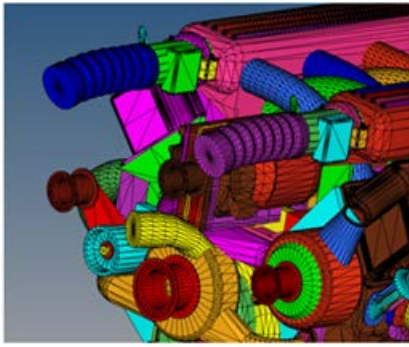
- Split/collapse edge



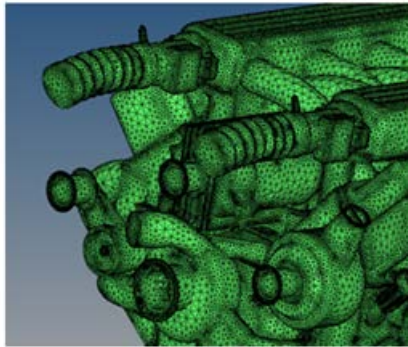
- Smooth: adjusts the node position. There are additional options to select anchor nodes for preservation during the smooth operation.
- Drag tria element: enables the movement of a tria to a more suitable location simply by dragging the element to the new location. An important capability for controlling the mesh aesthetics.
- Split quad element and combine tria element

Adaptive Wrapper

Adaptive wrapper provides a quick solution for dirty CAD and has a ability to do local refinements. It preserves features and can refine between components. The tool contains a utility to find leaks in the model and also contains automated hole or gap patcher.



Before



After

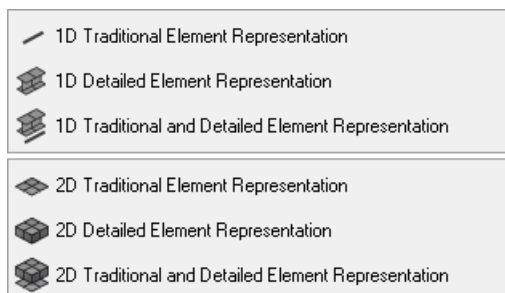
Resolved Issues

- Some steps in Topology cleanup process that ends up deleting one component during a BatchMesher run for some models
- Coarsen Mesh functionality incorrectly deletes a component for specific models
- Mesh lines do not match the surface edge line when using biasing in Automesh panel
- "Split in to hexas" option inside the solid elements subpanel of the Split panel is not working correctly in some cases due to wrong node association
- SPC constraints were created even though the "Create constraints" option was not selected in the Box trim dialog. This happens when meshing was performed prior to box trim run.
- Huge jump in tetra core element size with and without "Max tetra size" selection in tetramesh parameters subpanel or the Tetramesh panel
- "Place node" option in the Cleanup Tools section of the Quality Index panel was crashing the application
- Quality Index panel crashes the application if user zooms into the model using Ctrl + left mouse button and does a circle zoom
- Translation of components loses node association within solids. This was happening for inner nodes for some specific models
- Saving a HyperMesh database file crashes the application in some cases after using reducing tria element selection in Element Cleanup panel

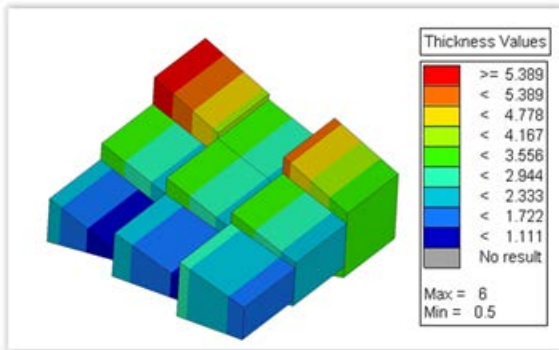
Graphics

Enhancements

- The detailed element representation has been split into separate controls; there is now control for only 1D elements or 2D elements. It is now possible now to see only the 1D elements in a 3D representation if desired.



- The detailed 2D element representation takes into account node thicknesses now too. In the past only the thickness from property and section cards have been taken into account.
- Within the Visualization options tab there is new control for load labels, and within the Display toolbar there is new control to alter the load vector visualization orientation. The load vector visualization can be defined in one of two ways, either the tip or the tail of the load vector is attached to the load application point.
- Graphics area screen capture; there is a new option within the Color options dialog to force the text, such as labels, legends and titles etc, to be black.
- Nodal thickness visualization is supported for RADIOSS, Abaqus, LS-DYNA and Nastran user profiles.



- A new 'by Assembly' Review mode is added to geometry visualization toolbar

Resolved Issues

- In certain situations bar3 elements were incorrectly drawn. The node order was incorrect for graphics only.
- In certain situations the 3D ply visualization has been wrong at the node, where two different plies were attached to each other.
- There was an inconsistent behavior for the "by face" selection of elements and surfaces. For surfaces the selection by face included mask surfaces, hence all surfaces were selected in certain situations. The by face selection no longer includes masked entities now.
- Selected nodes didn't appear as highlighted complete spheres.
- When using the element quality view selecting and deselecting the criteria to be shown in the legend caused HyperMesh to occasionally crash.
- HyperMesh crashed sometimes when deleting loads or load collectors which have been selected by window.
- Free lines in combination with the use of spherical clipping were wrongly displayed. Only if both ends of one line have been inside the sphere the display was correct.
- Surfaces in combination with the use of spherical clipping were wrongly displayed. Only if at least one vertex has been inside the sphere the display was correct.
- The visualization of beam sections wasn't always updated immediately in the graphics display when beam attributes have been updated.
- Entity selection using 'by block' is not working when the local system used to create the block is deleted
- Display becomes unresponsive when using Ctrl+ left mouse click to rotate a large model on Linux and MAC machines
- Surface pre-highlighting during node list selection was not working in the Surfaces panel
- Performance issue related to nodes selection on Linux platform
- While using the color by assembly the elements are colored with the assembly color but the geom is colored with the comp color
- Segmentation fault when you rotate a model while using toggle edge inside Quick Edit panel

- 3D element representation display was not working correctly at t-connection
- Elements normal display inside add subpanel of Interfaces panel was not working
- Segmentation error when you click off after selecting all systems inside Numbers panel
- Plot display turn on while using Show and Isolate for includes

Known Limitations

HyperWorks Graphical User Interface (GUI) products (HyperMesh, HyperView, HyperGraph, MotionView, etc), when used with Ultra High Definition (UHD) displays consisting of 1920x1200 or higher (for example, 3200x1800, 3840x2160 or 4096x2160), may result in graphical problems. Some GUI items may not be displayed correctly, some functionality such as image captures may not capture images correctly, and window selection may become an issue. HyperWorks 13.0 does not officially support these high screen resolutions. We suggest setting your resolution to 1920x1200 or lower to resolve or minimize these types of issues from occurring. Our development teams are aware of these problems and are working closely with the hardware vendors to find a quick resolution to the issue.

API

- A new set of APIs for performing entity creation (*createentity and *createentitysameas), entity update (*setvalue) and entity query (hm_getvalue) is officially released. These APIs replace many existing HyperMesh Tcl Query and Tcl Modify commands, and help to streamline the scripting and automation process.
- New hm_collision* APIs and functionality for performing detailed result query.
- Added the ability to query the minimum internal (hm_entityminid) and solver IDs (hm_entityminsolverid)
- APIs to convert a click in the graphics area to a 3D model coordinate (hm_viewproject) and vice-versa (hm_viewunproject)
- *createmark/*appendmark/hm_createmark/hm_appendmark have been enhanced with new selection methods "by cone", "visible", "by metadata name", "by metadata contains value", "by metadata equal to value", "by metadata greater than value", "by metadata less than value", "by metadata value range" and "by metadata type".
- See the API Programmer's Guide in the HyperMesh Reference Guide for full details on API updates/modifications.

12.0.115 Release Notes

New Features

Assembly Browser

- Performance has improved during the save PLMXML process for large BOMs.

CAD

- Support for UG NX 9.0 (native reader)
 - UG NX 9.0 now requires the license server to be defined using the SPLM_LICENSE_SERVER environment variable, while older NX versions still require the UGS_LICENSE_SERVER environment variable.
- Users can now identify specific parts to include/exclude during JT import using the @FilterByJTOpenParts option in the jt_reader.ini file

Connectors

- New realization is supported and has been updated to the existing configs:
 - RBE3 spider/wagonwheel bolt realization
-

Resolved Issues

API

- Fixed performance issues with hm_getentityvalue command
- *graphuserwindow_byXYZandR behavior has been corrected

Assembly Browser

- Resolved issue where thickness updates to property cards were not reflected in the Assembly browser
- When importing multiple representations pointing to the same material MID, multiple renumbered materials are no longer created
- Fixed several save PLMXML issues

CAD

- Addressed a few crashes and instances of incorrect data for JT import

Meshing

- Fixed issue with component hole detection where certain holes were not properly found
- Resolved several issues with MPC generation and hole patching for acoustic cavity meshing
- Resolved issue related to midsurface mesh application error while loading parameter and criteria file on Linux platform
- Resolved issue related to feature edges not deleted after midsurface mesh run
- Resolved segmentation error related to using abort button in Quality Index panel in an empty session

Graphics

- Resolved a node display issue
- Resolved a selection issue related to elements in a set that caused a session to hang

Connectors

- Hillock CBUSH stiffness values rectified in some cases
- Resolved application error while changing the elem filter options in FEAbsorb adhesive
- Resolved issue with connector lines export
- Resolved issue with Penta MIG connector realization with the correct orientation
- Resolved bolt realization failure issue for a tube section
- Resolved issue related to Hillock PBUSH property creation

Collaboration Tools

- Fixed the issue where files with German characters were not added correctly during the Add Folders and Files process.
- Resolved the issue with remote libraries where pre-defined workspaces were not visible.
- Resolved the issue where the application moved to the background each time new content was published.

12.0.113 Release Notes

Connectors

Bolt L1/L2 Graphical Cylinder with Vector

- Cylinder preview
- Vector preview
- Temporarily or permanent display
- Transparency changeable

Export Dynamic Bolt Vector Option

- For MCF export (version A) the dynamically identified bolt connector vector will be exported by default now

Link Groups now visible in Connector browser

- Properties are now available as connector links
- A special property view has been added to the connector browser
- Link groups are shown in the connector browser
- Details are given in message boxes
- All find and display mechanisms are supported for properties and groups as well
- Property link groups are exported in MCF

New RADIOSS and Dyna Connector Realizations

- Automatic exclusion of of special nodes during rigid bolt realization (HyperCrash logic)
- Hexa spot with contact and material (law 59) definition
- Seamweld and adhesive connector types supported via MCF
- Connector realization type specified in new mapping file for automatic realization
- Users' connector type selection limited to only thos available in mapping file
- MCF Properties available as columns in Connector browser

Enhanced MCF Support

- Import and Export of MCF supported
- Property links supported
- Ability to move connectors between MCF files

Connector Auto-realization

- Connectors realized automatically upon import

- Log file generated providing realization details
-

Module Enhancements

Current Module Support

- Set current module similar to current component behavior, from
 - Assembly Browser thru a context menu
 - From HM status bar

Current Module & current component automatically kept in sync

- Module-Entity Organization
 - Provides the ability to organize components, materials or properties from one module to another
 - Module Show Contents
 - View the components, materials, and properties owned by the selected Module
-

Assembly Browser Enhancements

Batch mesh integration UI enhancements

- Individual modules can now be meshed to specific mesh criteria and parameters; previously, all files had to be meshed to the same criteria
- Mesh criteria and parameter files can be mapped to values in the Assembly browser for automatic selection

Assembly browser filtering

- The Assembly browser will now provide advanced filtering tools to filter BOMs based on their properties, existence of a representation, etc.
-

Resolved Issues

- When renumbering there are instances of duplicate entity IDs being created. The most prevalent is duplicate element IDs. However, duplicate property IDs can also be created. Focusing on the element use case: the duplicate element IDs can be created once a renumber operation is complete followed by an operation that creates elements (such as split, or duplicate>>translate/rotate, or even mesh). The issue can occur with the following user profiles - Abaqus, Permas, Nastran, OptiStruct, Ls-Dyna, RADIOSS and Pamcrash.
 - Duplicate set id's are created as a result of duplicate>>translate/reflect operations on rigid elements. This issue affects the RADIOSS user profile.
-

Known Assembly Browser Limitations

- Show contents
 - There is currently no way to visualize the contents of the default master connection file. This leads to the risk that the user will lose connectors if they're in the default file but not moved to an MCF file that is a representation of a module
- Import from Assembly browser
 - If an error is encountered during importing of multiple representations, then the import process stops rather than continuing from the next representation in the list. Users can continue by selecting the remaining representations and importing again.

12.0.112 Release Notes

User Interface

- Bolding of the browser tab text has been added to provide feedback as to which is the active tab.
 - Added a partition in the browser to indicate a separation in the controls between the global display controls and the action modes (selector, show/hide, isolate).
 - Added toolbar tool tip help when mousing over the toolbar handle.
 - Reorganized the Mesh>Edit pulldown menu.
-

Connectors

- A new realization type for the Dyna user profile has been introduced, the ACM (shell gap + coating).
 - This realization creates a round hexa pattern between shell flanges by creating congruent meshes.
 - The user can chose between different hexa patterns and define the diameter. Additionally the creation of a washer layer can be requested.
 - For each hexa pattern a keyword *DEFINE_HEX_SPOTWELD_ASSEMBLY_<# of hexas> is created.
-

Shell to Solid Conversion

A third method of creating Solid composite layers from shell composite definition for Abaqus is supported. In this method users can group ply layers using a dummy plies and the software will create a single solid for the group of plies separated by the dummy layers. With the addition of this method, users could create single solid for all the plies or a single solid to represent each ply or multiple solids for grouping of plies.

Line Loads

Ability to assign total or distributed loads to nodes on a geometry line or a list of nodes is added, and then can automatically calculate the nodal loads.

Improved Performance

Improved performance of HM FBD tools for OP2 results and added support for XDB results.

Resolved Issues

- Segmentation error with OSSmooth
- Segmentation error when splitting surfaces
- Segmentation error when tetrameshing
- Long selection times when picking nodes by window on Linux
- Display of normals not working correctly in the Interfaces panel
- FE overwrite behavior issue when importing properties in OptiStruct and Ls-Dyna solver interfaces
- Resolved several file specific issues related to the JT, CREO2.0 CAD readers
- CFD Meshing

- Selecting boundaries with quad elements as “with BL float” fails to generate mesh
- Using the selection “with BL float” and rejecting the tetramesh generates free edges
- Using remesh including a refinement zone, the resulting remeshed region doesn’t follow the shape of the refinement zones
- Browser
 - FE/Geometry display/filter mode wasn’t working correctly in certain cases
 - “Show” operation was returning the plots. Show behavior is now consistent with Hide and Isolate
- Ansys Interface
 - Issue with Constraint Equations where equations with thermal labels (DOF) were not handled by HyperMesh in earlier versions has been fixed now
 - Issue with export of load steps where loads applied on same node in same direction were not combined is now fixed
- OptiStruct to ANSYS conversion
 - Defect where material properties to the components were not getting assigned after conversion has been fixed
- Abaqus to OptiStruct Conversion
 - Conversion of Abaqus *MPC PIN to OptiStruct RBE2 without DOF values issue has been corrected
- OptiStruct Interface
 - HyperMesh error message due to renumbering of DTABLE entries has been fixed
- Abaqus Interface
 - Defect while exporting an input deck with a MPC Beam converted into a Kinematic or Distribution coupling
- LS-DYNA interface
 - Issue with conversion of spotwelds from Nastran to LS-DYNA has been fixed
 - Issue with reading parameterized *ALE_REFERENCE_GEOMETRY_BIRTH_RDT keyword has been fixed
 - Support of -2 and -1 options for ELFORM in *SECTION_SOLID
 - False error message during import of *CONSTRAINED_EXTRA_NODES_NODE/SET has been removed
- When renumbering there are instances of duplicate entity ID’s being created. The most prevalent is duplicate element ID’s, however, duplicate property ID’s can be created. Focusing on the element use case; the duplicate element ID’s can be created once a renumber operation is complete and then performing an operation which creates elements, such as split, or duplicate>>translate/rotate, or even mesh. The issue can occur with the following user profiles - Abaqus, Permas, Nastran, OptiStruct, Ls-Dyna, RADIOSS and Pam-Crash.
- Duplicate set V’s are created as a result of duplicate>>translate/reflect operations on rigid elements. This issue effects the RADIOSS user profile.
- OptiStruct and Nastran Interface: resolved a crash when importing a file with mixed HyperMesh and Patran comments.

Known Issues

A crash can be encountered if either Catia, ProE, ACIS, STEP, SolidWorks, or non-native UG reader is invoked at least once, and, within the same session, the native Parasolid reader is then invoked - the crash occurs during this secondary import operation. If the order of these two operations is reverted, the crash does not occur. The workaround is to import the parasolid file with parasolid reader within a brand new HyperMesh session.

12.0.110 Release Notes

General

ID Management

For Ls-Dyna and Nastran user profiles, a new ID management capability has been added. The toolset enables complete control for all ID rules and renumbering operations based on include files. The renumbering can occur when creating a new model or resolving issues with a legacy model. There are a number of advanced operations such as locking of specific ID's, the ability to exclude certain entities from any renumbering operation, plus much more. There are several correction renumbering options if a ID overflow has occurred. There are also user definable ID rules for any newly created entity.

Composites

- Laminate realization has been enhanced to support core sample validation
- Zone based Nastran, Abaqus and Ansys composite property creation has been added
- Laminate editing functions have been enhanced to support copy, insert and modify ply layup and import and export of ply stack data from Excel and CATIA
- The ply realization option has been enhanced by adding additional search methods and sample points for boundary recovery
- New ply visualization tools are introduced to separate individual plies and compare them with the geometry ply with the resulting elements sets
- Exporting ply and stack up data from HyperMesh to CATIA is added either using CPD or Fibersim
- Converting zone-based composite models to ply-based models is enhanced

Resolved Issues

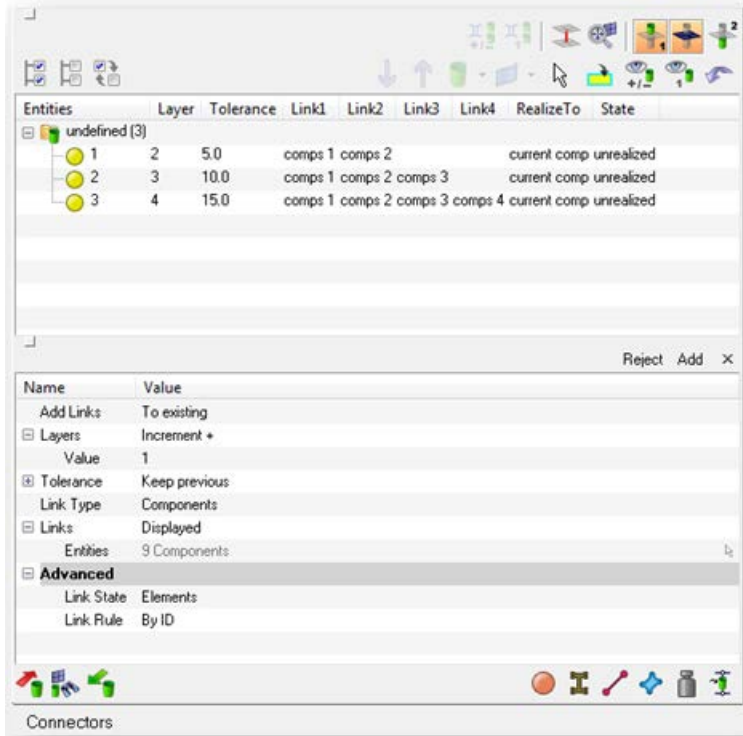
- Robustness of CPD cad import and ply realization has been vastly improved
- Robustness of Fibersim import and ply realization has been improved
- Display of ply and elements associated to ply has been enhanced

Connectors

Enhanced Add Link Capability in the Connector Browser

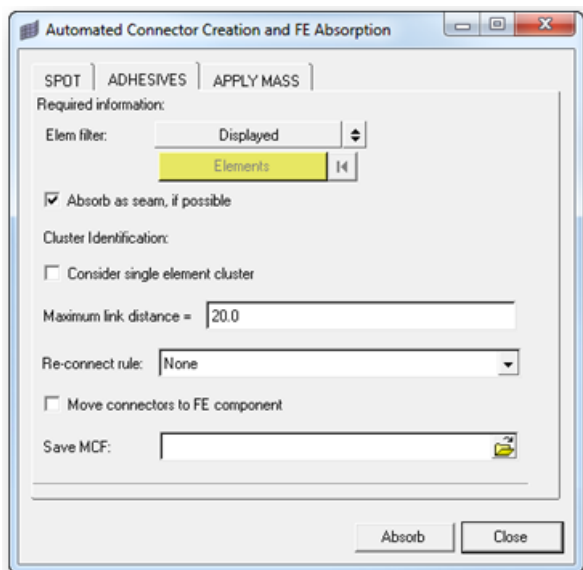
The **add link** functions in the **Connector Browser** include further enhancements of former add links capabilities. It is now possible to:

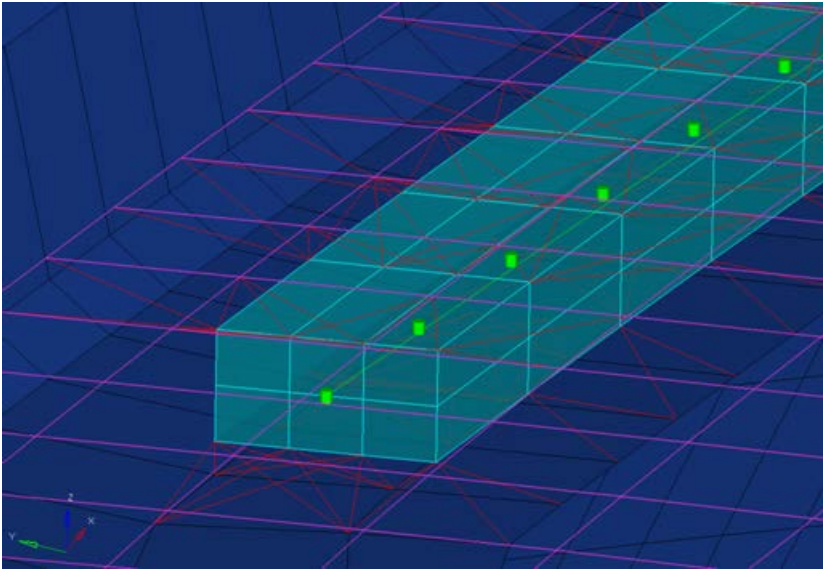
- Update the connector links of multiple connectors at once by preserving individual settings that were previously defined, such as tolerance and number of layer.
- Define the number of layers by the tolerance, instead of having to predefine it.
- Add connector links from scratch by removing any links that were previously defined, and then starting a search for new links to add on a cleared connector.
- Reject capabilities have now been added to the add link tool.



Connections of Type Hexa Adhesive can be Absorbed as Seam Connectors Now

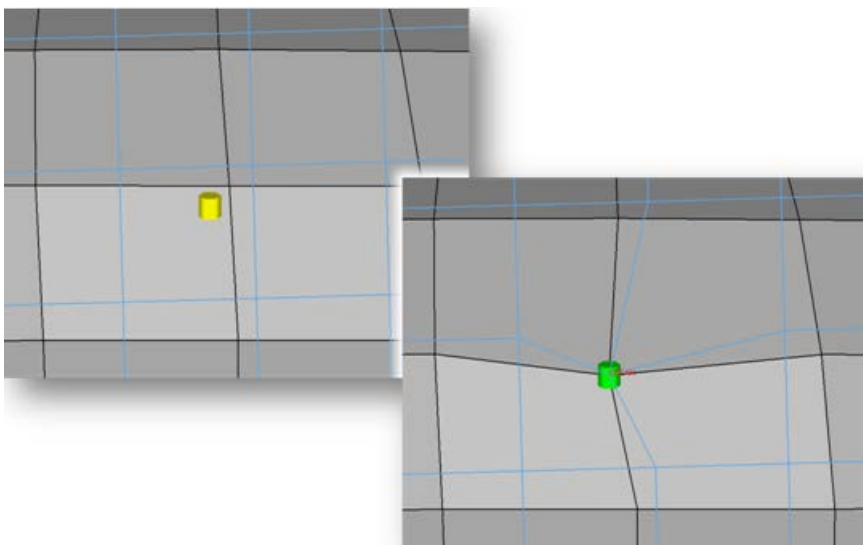
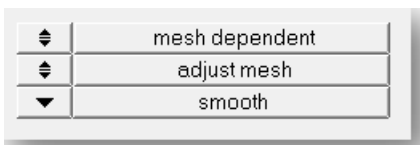
In the **Absorption dialog > Adhesive tab**, a new checkbox **"Absorb as seam, if possible"** is now available. It is activated by default, and allows hexa adhesives to be absorbed as seam connectors as long as it has a regular pattern from exclusively hexa elements. If the hexa adhesive does not have a regular pattern from exclusively hexa elements, it will be automatically absorbed as an area connector. During the absorption the element pattern, dimensions and tolerance are detected and stored on the connector.





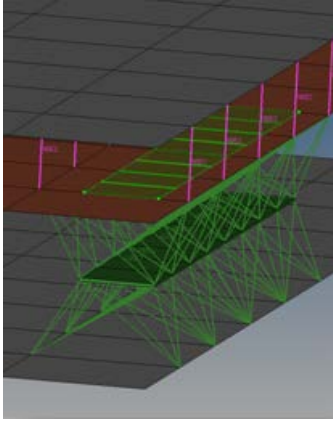
A New Mesh Dependent Realization Approach "Smooth" has been Added for the Spot Realization

With the **smooth** option, the mesh will be adjusted in such a way that the closest node to each layer is moved into the position of the projection point. The advantage of this option is that neither the element IDs nor the node IDs will be changed, and the element topology will remain the same.



New Area Connector Realization Type "Adhesive Hemming"

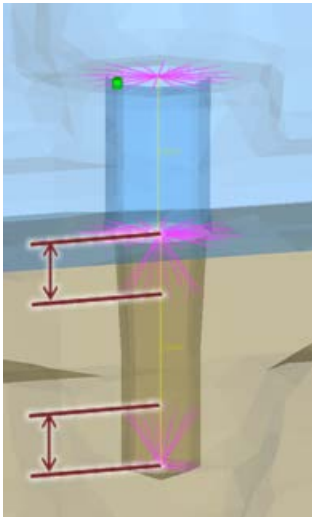
A new area connector realization type, adhesive-hemming, is now available for Nastran and OptiStruct. This realization type is used for modeling roll hemmings, where the outer shell is bent around the inner shell. The inner shell is connected to the outer shell on one side with simple hexa adhesive, and the other side is connected with RBE2 elements. A definable orientation node decides which side the hexa adhesive should be used. This seam realization type is capable of connecting three layers that contains two components.



New Bolt Connector Realization Type "bolt (threaded step hole)"

A new bolt connector realization type, bolt (threaded step hole), is now available for Nastran and OptiStruct.

This realization has the capability of connecting two solids through holes, or connecting one solid through a hole with a solid blind hole. A thread length can be defined to define the dimensions of the rigid elements connecting the bolt shaft models as a bar.

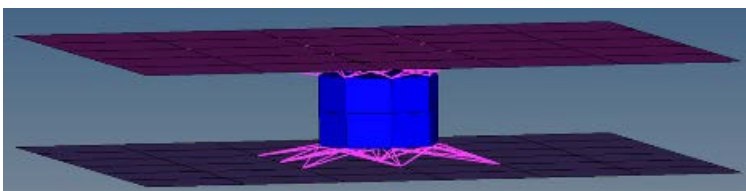


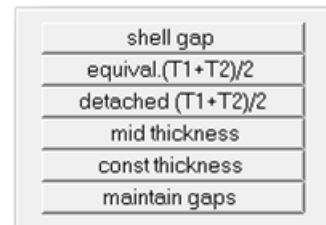
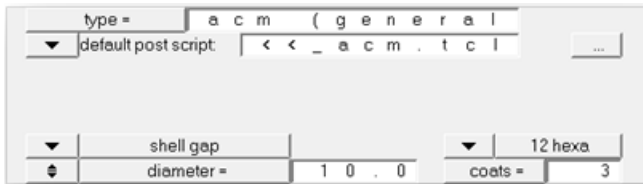
This realization has the capability to connect two solid through holes or one solid through hole with a solid blind hole. A thread length can be defined to define the dimensions of the rigid elements connecting the bolt shaft models as bar.

New Spot Connector Realization Type ACM (General)

A new spot connector realization type, acm (general), is now available for Nastran and OptiStruct.

This realization type consolidates several ACM definitions into one general, flexible ACM definition. Besides mid thickness, constant thickness and maintain gaps, the definition of several coats with different hexa patterns is available.





Other Enhancements

- An updated version (2.0) of the XML connector file has been introduced. This new format can be read as well as written now. It covers the latest master connection file version 1.1.
- The vip file format is now consistent for both import and export. There is no longer an eight column fixed format, and the link IDs are always separated by space.
- Absorption of the realization type acm (shell gap + coating) is now available. The hexa pattern and diameter are detected and stored on the connector.
- For the absorption of the Dyna realization type Mat100 (hexa), the check for the correct material can now be switched off to allow the absorption of incomplete hexa patterns.

Resolved Issues

- The Abaqus realization type fastener assigned the same *FASTENER PROPERTY to fasteners created between the same components but with different diameters.
- In certain cases the re-realization of the realization type acm (shell gap + coating) with exact same settings failed.
- In certain scenarios an empty "auto1" component remained after realization.
- Certain clear boltholes were not able to be detected, resulting in failed bolt realizations.
- The coordinate system for the Hilock realization has been changed from a left handed one into a right handed one.
- During seam partition, all of the selected seams were unrealized, as well as the seams that were not partitioned.
- The context menu in the upper part of the Connector Browser did not display properly.
- The configurations of the lower part of the Connector Browser was always set back to the default.

Geometry

Enhancements

- LCS option for creating surface by drag along vector
- Adding fixed points on ends of surf lines if fixed points are deleted
- Include all points from the original line and project them on the offset line
- Improvement trim with plane of surface edit when surfaces are modified by releasing points or toggling edges
- Function to add fix points at inflection point on the free edges surfaces
- Option detach solid from solid
- Creating solid by dragging surface such the generator surface in the middle (+1/2 and -1/2)

Resolved Issues

- Midsurface, sort function delete the elements associated midsurface

- Surface and Solids creation panel display problems
- Solid is displayed as collapsed when change to "Auto" as level of refinement
- hm_measureshortestdistance2 command cannot search nearest solid in some case
- Auto cleanup – crashes HM are fixed
- Detach solids selectively
- Robustness improvement to IGES, JT, CATIA and Parasolid and UG import are added
- Many bugs related reject operations are fixed in the Geometry panel
- Creating smooth lines using nodes is improved
- Tangent line creation option is improved
- Many improvement to midsurface creation are added
- Many solid edit options are improved

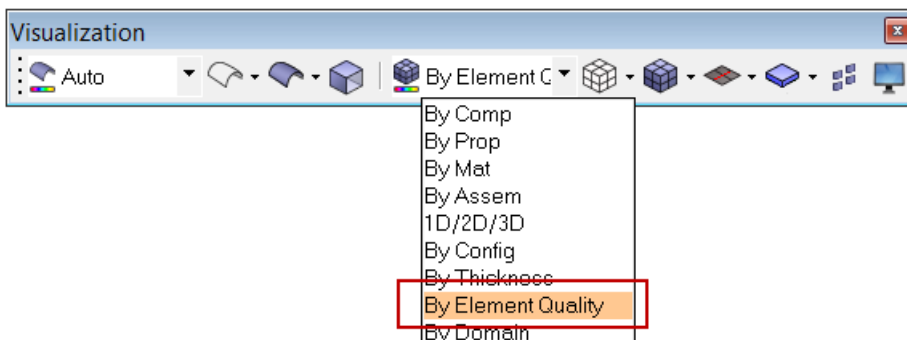
Graphics

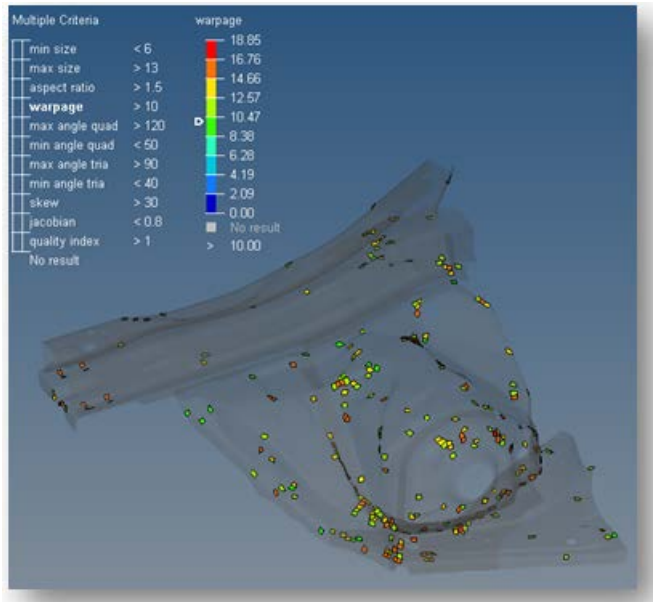
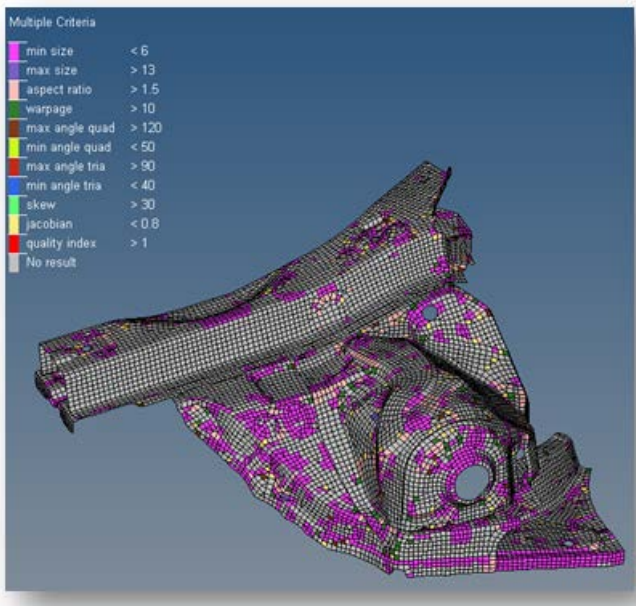
Performance Improvement

Major performance improvements for handling of geometry models. Interactive dynamic operations such as pan, zoom, and rotate are substantially faster than the previous release. The handling of geometry models feels more lightweight reducing any perceived lag when moving the model in the graphics area. Faceting time has also been improved.

New "By Element Quality" Visualization Mode

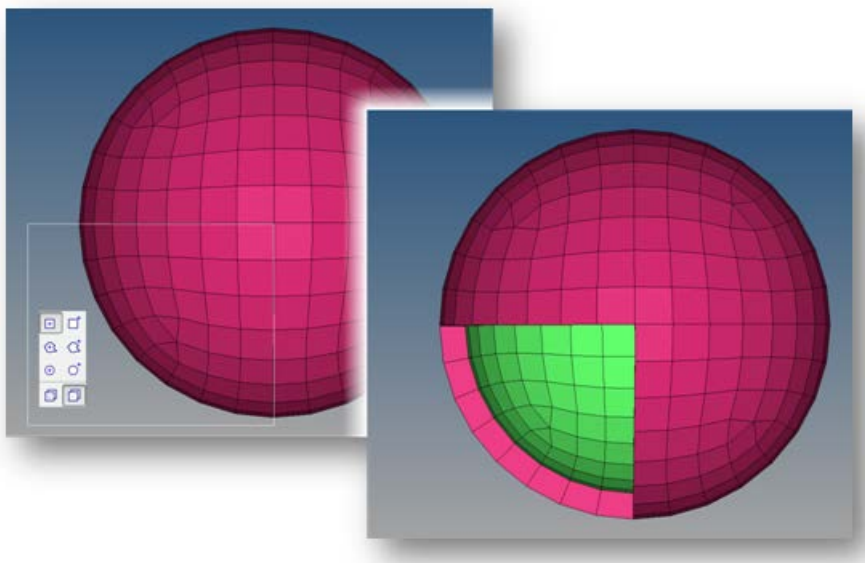
This new visualization mode enables a perpetual element quality visualization representation regardless of which operations are being used. The visualization mode allows you to move the mesh, for example by morphing or translating nodes/elements, which in turn dynamically updates the element quality graphics display. The visualization mode allows the check of multiple element criteria at once using a particular color or there is also a secondary mode enabling a more detailed review of a specific element criteria, for example max size. Elements of either specific or no interest can easily be removed from the display via filtering enabling a quick review of those elements that fail a criteria.





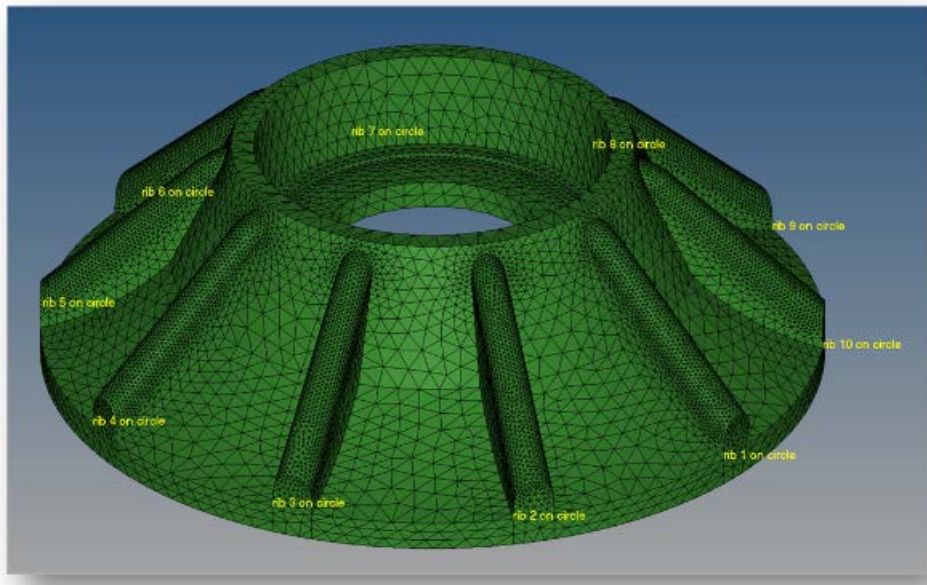
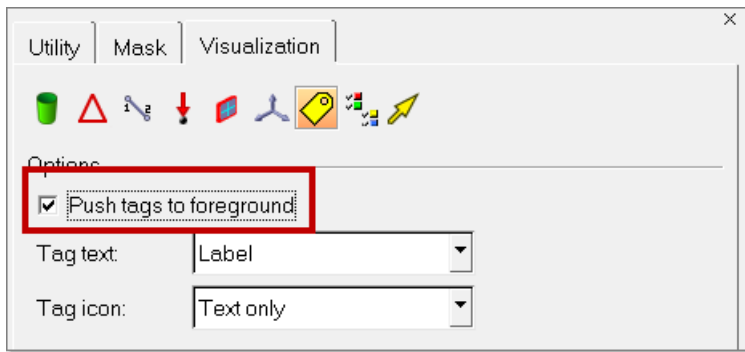
New Element "By Visible" Window Selection

The By Visible selection mode is enabled via the window selection menu (shift + RH mouse button). At the bottom of the menu, there is a **visible** option. Once enabled, it will work for rectangular, polygon and circle selection. By Visible selection is only supported for elements.



Tag Enhancements

- Within the graphics area, tags are never seen *partially* anymore. They are fully viewable, or fully hidden. This depends on the visibility of the entity the tag is attached to.
- Tag labels can be created from the entity ID.



Center of Rotation Behavior

Within the Options > Graphics panel, there is a new option "rotate about mode". The option defines whether the model will be rotated around center of the screen or the center of the model. The option is enabled by default which will retain the rotate behavior as per earlier releases. If the option is disabled, the rotation will be about the center of the screen (not model) which is the same behavior seen in HyperView.

Resolved Issues

- There were different scenarios, where the model was flickering during rotation.
- Certain model combinations lead to incorrectly colored mesh lines for the mesh preview in the **Automesh** panel. Green mesh lines were shown on green element faces, which made it hard to see the mesh.
- Many smaller issues in combination with the composite view have been resolved.

HyperBeam

HyperBeam GUI Enhancements

- **Undo/Redo**
Four buttons were added to the toolbar at the top: **undo all**, **undo last**, **redo last**, and **redo all**. These allow you to move backward or forward through any changes you have applied to the currently displayed beam section (dimension changes, part additions, thickness changes, etc.). If you switch to a new beam the undo/redo list is cleared for the previous beam. The **reset** button in the **Parameter Definition** window (lower left) was removed as it was no longer needed.

- **Moment of Inertia** tool
A button was added to the toolbar named **find moments at mouse location**. Clicking the button launches an interactive mode; clicking on the graphics screen will display a window showing the Y and Z coordinates of the point selected, the Y and Z offsets from the shear center and centroid, and the moments of inertia I_{yy} , I_{zz} , and I_{yz} for the section relative to the selected point. This is useful for determining beam element offsets or making custom beams. Clicking **OK** removes the window and allows you to select another point.
- **Ruler** tool
A button was added to the toolbar named **measure section with ruler**. This button launches an interactive mode where clicking on the graphics screen two or more times creates a line similar to the way a shell section is created. If two points are selected the dy and dz distances between the points are shown as well as the total length of the line. If three or more points are selected only the total length of the line is shown. Right-clicking removes the previously selected point.
- **Scale option**
Right clicking on any beam section in the browser will reveal the **Scale** option. This displays a window allowing you to scale the entire beam section by a given scale factor.
- New magenta + symbol drawn in the HyperBeam GUI on all sections where the beam element axis is located.
- Added check for disconnected parts and error message for shell beam sections.
- **Import CSV**
A new option called **Import CSV** was added under the File menu. This option will read HyperBeam's CSV format and create new shell beamsections. The Import CSV and Export CSV options allow you to read and write to HyperBeam without using the .hm file.

New Beam Sections

- Cross (Nastran/OptiStruct/RADIOSS, LS-Dyna)
- Asym-I (Permas)

HyperMorph

New Features

- Graphic manipulators under Morph and Freehand panels for easy setup of interactive morphing
- "Skin only" option for morph volumes for faster morphing of large models especially in CFD
- "Fit to target" as a projection option while mapping domains, mesh, or morph volume faces to surfaces or a mesh for better description of boundaries
- "Matching" and "layer" constraint types to improve the quality of CFD boundary layers
- Simultaneous shape checking and smoothing to produce better element quality during optimization

Graphics Enhancements

- Different colors can be set based on domain type
- Color by domains option added in "Element Color Mode" selector
- Domain icons can be turned on and off from graphics preferences

Performance and Mesh Quality Enhancements

- Improved large domain solver for better element quality
- Step size option for interactive morphing so that the manipulator, handles, and nodes for freehand morphing only translate or rotate by specified increments

- “Autofix” option to apply large domain solver when small domains get folded
- Several new checks under auto quality check such as, 2D - squish, equia skew, taper, 3D - squish, equia skew
- Morph volume quality is checked after dragging
- New “unsquish” as smoothing option
- Several mid-node correction options such as hold current, fix midpoint, curve edges, curve faces
- Smoothed normals and “CFD corners” as projection options for freehand and map to geom panels to smoothly blend normal directions around sharp corners

Miscellaneous Enhancements

- Undo/redo list can be optionally saved with the file
- Nodes, handles, and shapes may be selected “by morph vols”
- Sculpting with 3D elements for tool as well as any assortment of 2D elements
- Local system selection via system collector under morph and freehand panels
- Internal partitioning of faces is optional for 3D domains
- When 3D domains are merged, customized 2D domains on their faces are preserved
- Convert shape extended to morph volumes
- Sets can be created for all morphing entities

Meshing

Box Trim

The Box Trim utility has additional mesh controls; the utility is able to remesh around the trimmed boundary ensuring a high quality quad dominated mesh. There are several advanced controls to specify the mesh size using a criteria file, other advanced controls include the possibility to exclude patches of mesh that serve no purpose after box trim – one or more parts could be trimmed leaving patches of elements, these can be removed.

Quality Index Panel - Cleanup Tools Enhancements

- Interactive nodes/elements modifications are now enabled to allow use of mask-related toolbars to hide/unhide elements.
- 2nd order elements are supported: middle edge nodes of 2nd order elements can be viewed and modified/optimized together with corner nodes

Automesh Panel Updates

- Enabled element remesh with different element size of selected area with size transition inside or outside of remeshing area
- Remeshing with “feature edges” option allows update of 1D elements after meshing
- Revert biasing direction by clicking on edge

Newly Added Tools

- **Element offset panel:** added distributed offset to assign different offset value on selected nodes for “CFD corners” method
- **Align Node > use ratio** is a new option which projects any selected node in between the two selected nodes a predefined ratio.

- **Min Size** check and fix function added in Tetra Mesh Optimization functionality.
- **2D & 3D drag**: added new capability to organize the mesh into original or current component.
- **CFD Mesh**: new option added to create a new component and organize the solid mesh when “simple select” and the “comp per volume” option is unchecked.
- **Second order Abaqus element checks are supported**: at corner and mid-points, and, at higher order integration points

Resolved Issues

- Added direct access to the param and criteria help from within the param and criteria editor
- Fixed issue with *shadingquality() hm.cfg command
- Add washer layer creation remesh issue is fixed
- Trim hole removes original elements correctly
- When in the meshing edit mode (secondary panel) it was difficult to see the editable values as they may be obscured by the mesh in the foreground. The editable values, such as node density or biasing, are now viewable.
- Crash using Quality Index > holes and washers > circumferential option
- Crash when in Quality Index panel and deleting a component from the browser
- Mid-nodes of hole and washer layer now maintain correct curvature
- Overlapping elements were created when moving a hole radially
- Solid Map did not create elements if the option “along geom: surfs” was used
- Performance improvements for morphing by volume
- Second order hexa to tetramesh splitting issue is fixed

HyperMesh APIs

hm_getconnected2delems

Returns lists of 2D shell elements and nodes that form T-connected, L-connected, Butt-connected or Overlap connected welds at component boundaries.

Direction and Plane Selection

APIs `hm_getdirectionpanel` and `hm_getplanepanel` have been added to allow users to define a direction/vector or a plane from within a script, using the standard panel vector and plane selector widgets.

Data Names for Principal Axes

Data names have been added for querying the principal axis values for components and elements, both about the global origin as well as the entity COG.

Resolved Issues

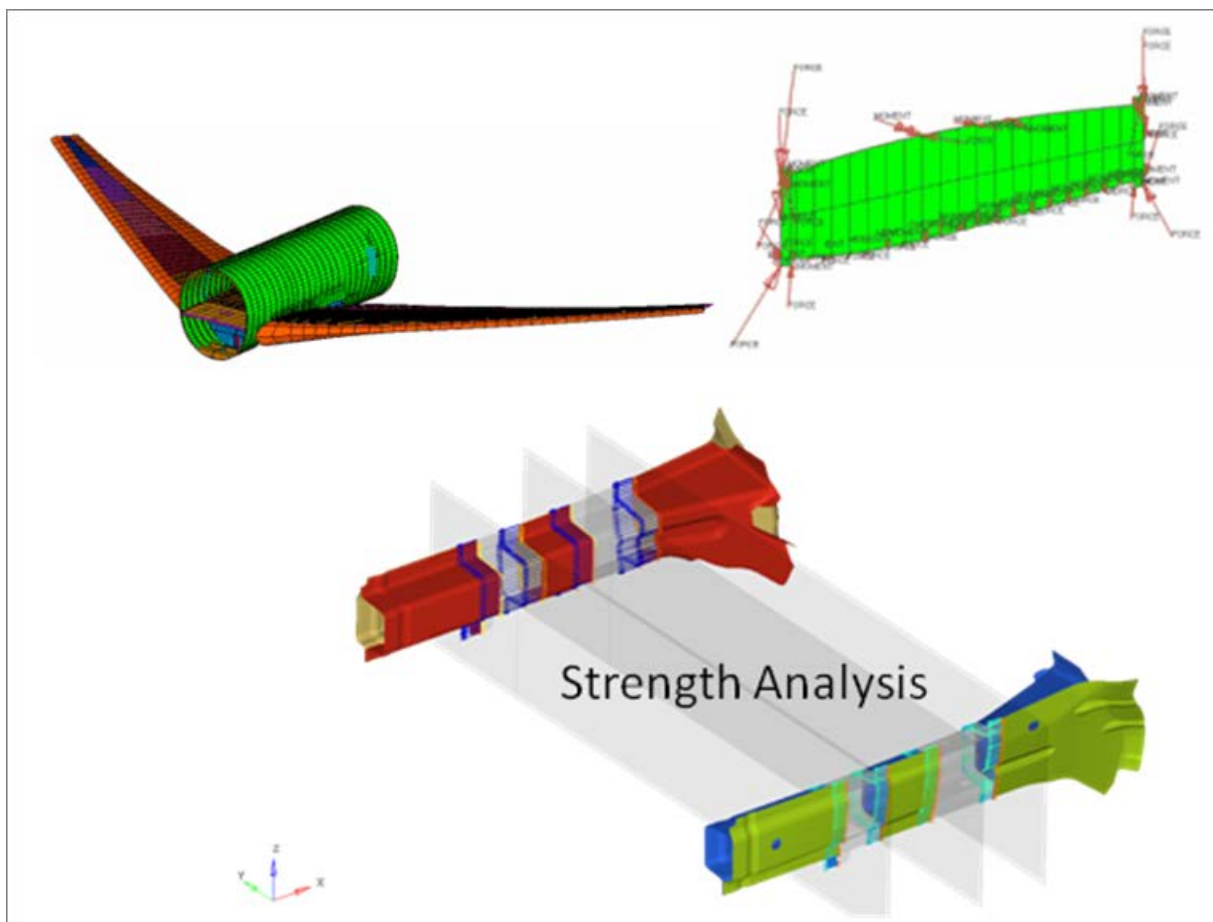
- `hm_blockredraw` now properly blocks all graphics operations while it is enabled

HyperView 13.0 Release Notes

13.0.0 Release Notes

Free Body Forces and Summation on Sections

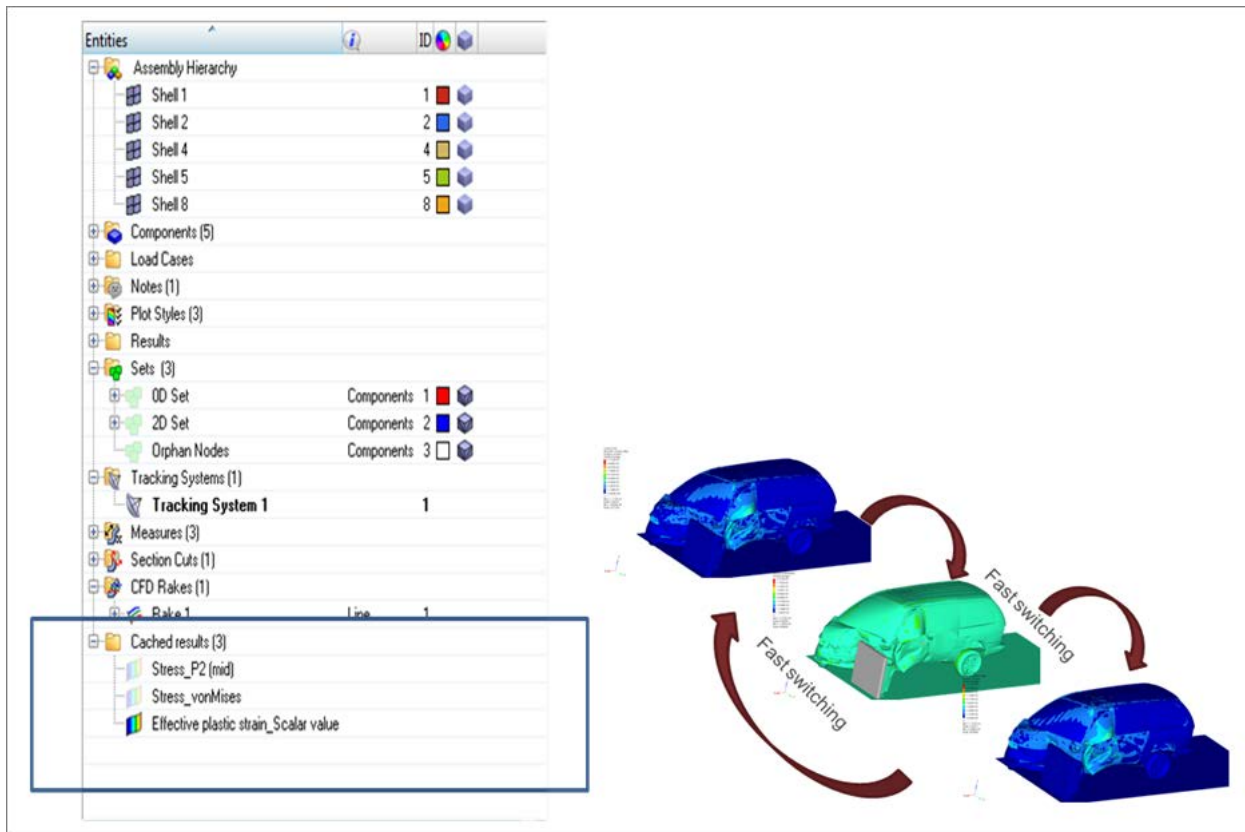
Free Body Diagram (FBD) tool enables analysis of free body forces and resultants on different sections of the model. Frequently used in design and analysis of structures, the FBD tool allows extraction of forces and moments for setting up a breakout model and/or sub-modeling analysis schemes, obtaining total loads acting on the section for strength analysis or for checking the equilibrium of sections. Users can define multiple sections that can be simultaneously plotted with either the free body or resultant loads. The forces on the section can be resolved in any user defined coordinate system. A further breakdown of loads as reaction loads, applied loads, or total loads on the sections is possible for both plot types. The results are plotted in the graphics area for the current loadcase that is active and the plot is updated as the loadcase is changed. Additionally, a summary table can be generated for multiple sections across numerous loadcases. The sorting columns in the summary table help quickly rank the significant loadcases for any min/max value of the resultant loads or the components. The summary table can be exported to a plain text file or an FBD file. The FBD file is recognized in HyperGraph for generating Potato plots, and Shear Force and Bending Moment diagrams.



Caching Results

Caching results makes it possible to preload the contoured results into memory and access them at any time in the session. It is relevant to users who have large memory (RAM) at their disposal and are dealing with big models that take time on plotting results. Caching helps to better utilize unused memory and

allows switching between results quickly. Any contoured result can be cached in to memory the first time, and retrieved subsequently in the session without any delays. These cached results appear in the browser tree in the order in which they are created. Users can contour the cached result instantly by clicking on the icon next to the name, similar to other result plotting from the browser. Any undesired cached result can be deleted to reduce the memory footprint.

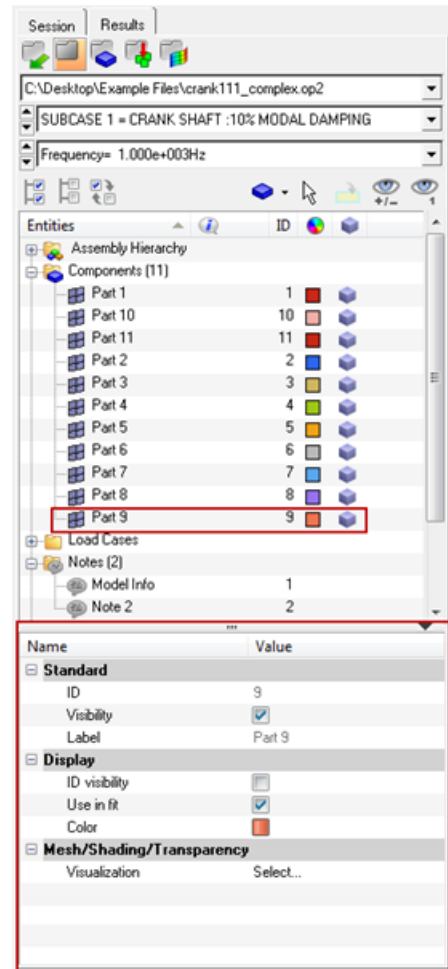


Entity Editor

Entity Editor is an enhancement to the browsers, allowing users to review and edit different attributes of the entities. Supported entities are components, sets, section cuts, measures, and notes. A preview pane at the bottom of the browser displays the attributes of each of the supported entities as they are highlighted in the tree. The attributes can be updated in the Entity Editor pane without going to any panels. Most of the operations performed in different panels is accessible now directly from the Entity Editor area. Display visibility, colors, fonts, draw modes are some of the examples of attributes that can be changed.

In addition to reviewing and updating in one place, the Entity Editor also provides additional functionality to do multiple updates at once. Multiple notes or measures or components can be selected in the browser tree and their attributes can be updated by changing the value in the Entity Editor name value table. If the attributes are different across entities, a hash mark (#) is shown indicating that not all entities have the same value.

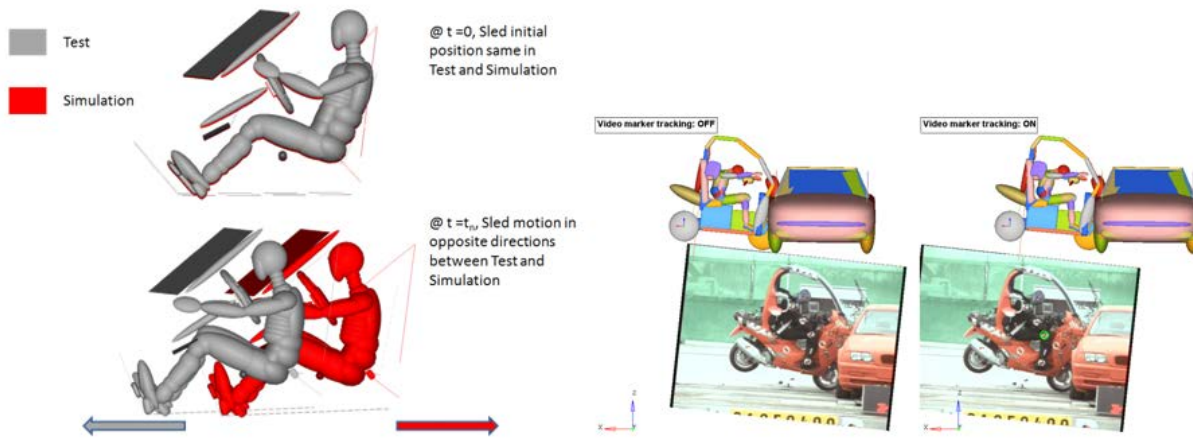
The Entity Editor makes a simple learning and a consistent experience for HyperWorks users, as the behavior is identical to the functionality introduced in HyperMesh. It is available already in HyperGraph for editing attributes of different entities.



Video Overlay - Marker Based Tracking

Multiple enhancements are made to the Video Overlay functionality, the most significant ones being the tracker based adjustment of image planes for correlating test and simulation when the set ups are different, as moving in opposite directions. A 3D image plane can be adjusted in the model space such that the relative position of a tracked point on the video remains the same with respect to the model. A new tracking option called Video Tracking is added under the Model tab allowing the user to select a reference point, and calculate its position across all frames. The image plane movement is then adjusted to maintain the same relative position of the point with respect to the model. 3D image planes also have an option of maintaining the aspect ratio when the size is adjusted using the manipulators.

A couple of automatic time synchronization modes are introduced to automatically adjust the scale and offsets between frames of the video and timesteps of the model. In case of overlaid models in a window the reference model to which auto sync modes are applied can be specified. The label field of the image plane tab has a drop down control that allows switching between the image planes without going back to the main browser view.



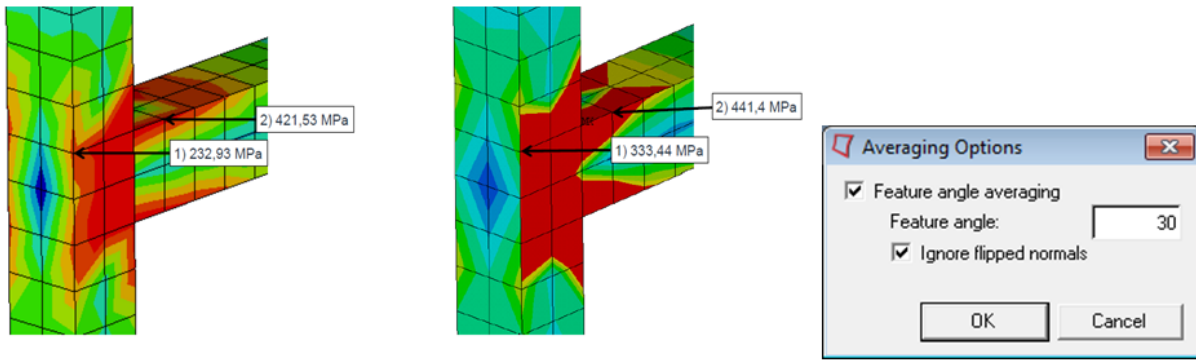
Marker Tracking

The Marker Tracking feature in MediaView allows the user to track any kind of markers or objects in MediaView as the video is animated. This provides numerical data which can be displayed and animated in HyperGraph. Manipulation of the tracking data by applying units or other math operations allows the comparison of tests and simulations. Markers could be added via mouse or API. Graphical options like show/hide tracking lines and marker IDs or show tracking markers and traces only makes it easy for the user to analyze the testing video.



Averaging Methods - By Feature Angle, Max of Corner

Feature angle based averaging of elemental results is available for contoured and iso-surface generation of results. Within a given part, the averaging happens for elements that are bound by a user specified feature angle as a threshold. This new option aids in evaluating stress distributions at T-joints and other sharp edges more accurately. Users have the option to ignore any modeling inconsistencies of flipped normals between adjacent elements when applying this averaging method.

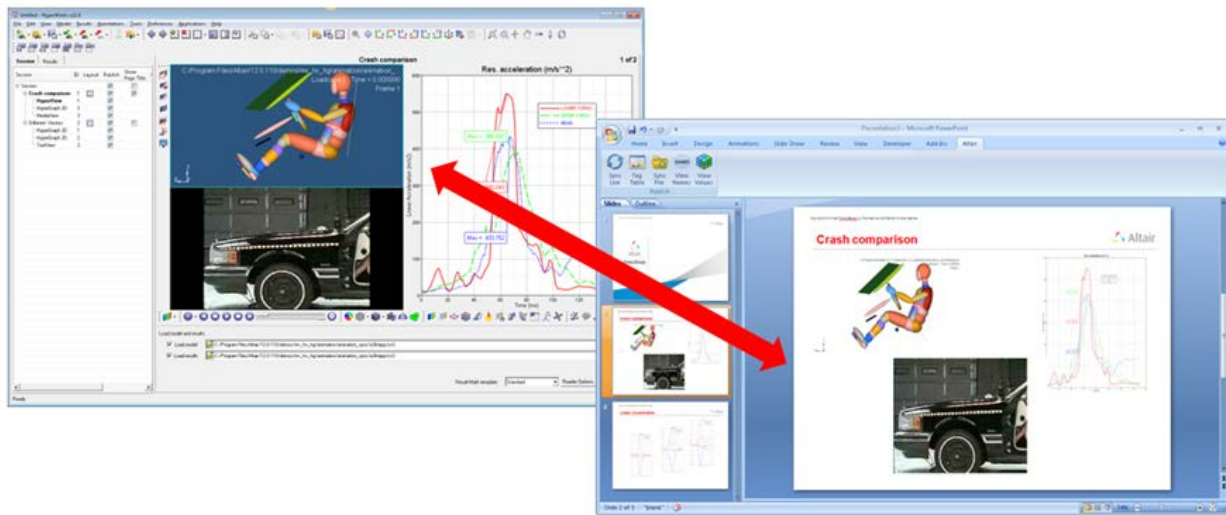


Courtesy of MAN Truck & Bus AG

For datatypes that have corner results, min/max/extreme of all corners within the element can be computed by using new averaging methods, "Min of Corners", "Max of Corners", "Extreme of Corners". The entire element shows the min/max/extreme of the corner values, which can represent a worst case scenario for aggregation of results.

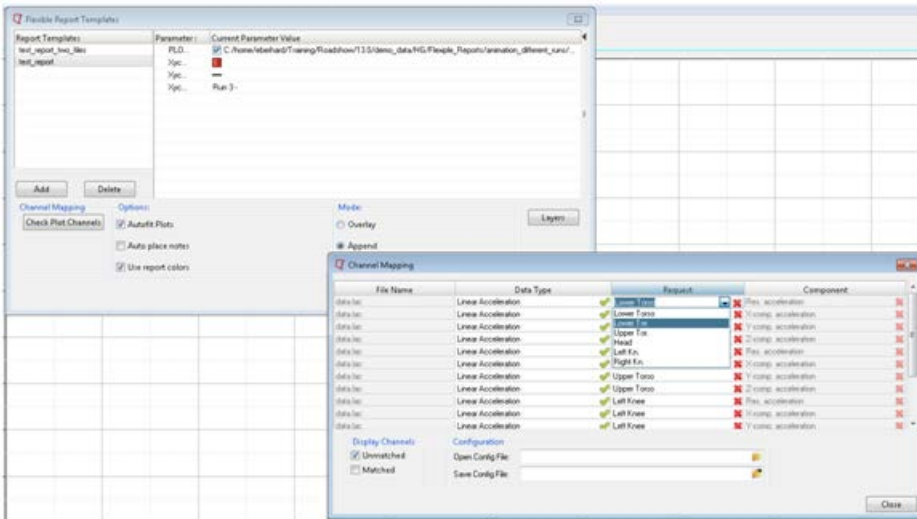
PowerPoint Publishing with Live Synchronization

The new HyperWorks Desktop (HWD) PowerPoint Publishing feature helps to reduce the work and effort in generating and updating Microsoft PowerPoint (PPT) Presentations from HWD sessions. This goal could be achieved by establishing a live link between HWD and PPT. The synchronization could be triggered either from HWD or PPT using a PPT Plug-in. Changes made in the PowerPoint PPT presentation will be preserved when new runs will be post processed after updating your session e.g. using Report Templates. The user can predefine the PowerPoint layout for the first publishing using PowerPoint Master Templates. The visualization of broken connections in PPT allows easy debugging if the PPT presentation will not be updated correctly. All HWD export formats are supported: Anim. GIF, AVI, H3D, PNG, JPEG, BMP, TIF. Dynamical Linking of HyperGraph notes with PPT allow to update text fields with new results. Live links are supported on Windows, Linux will be supported by data exchange over the file system using the PPT Plug-in.



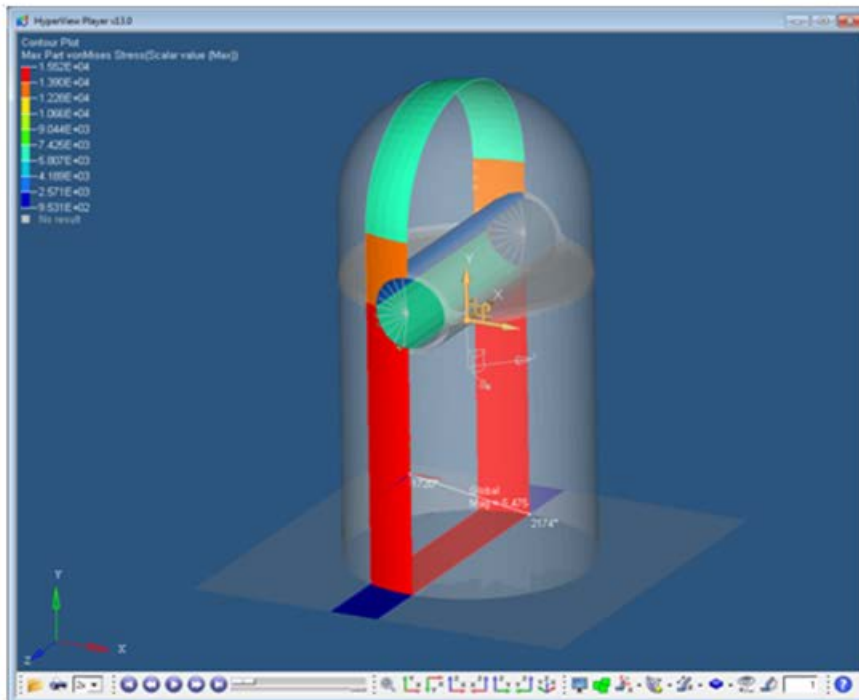
Flexible Report Templates

The new easy-to-use report Flexible Report panel replaces the Report Panel from earlier HyperWorks Desktop versions. It makes reports more flexible by giving the user advanced options to handle plotting channels during the import. More user-friendly color and style settings are possible for parameters created with the Parameter browser.



HyperView Player

HyperView Player has a new set of important functionality for interrogating the model. Measures, Tracking Systems, Section Cuts can be temporarily created in the player in addition to accessing the ones published in the H3D. All sets that are captured in the H3D file are also accessible to perform Show/Hide/Isolate tasks. A progress bar updates the status of loading the model and animating all steps giving useful feedback to the user. Deformation can be scaled to magnify the shape for clearer animation.



Engineering Solutions

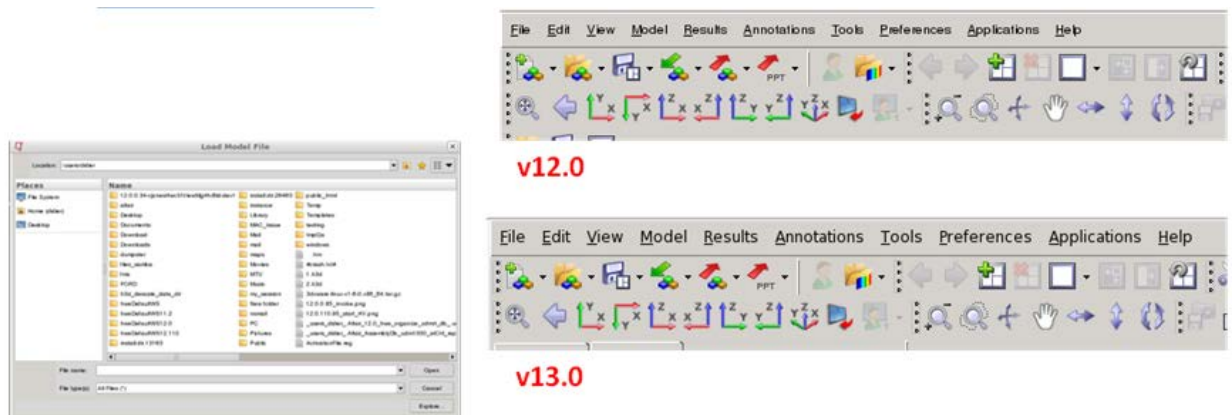
- NVH: Model and Test correlation tool MAC and CoMAC is added
- Crash: Diagnostic and model compare tools new, global control of data from browser, and flexible report templates for mapping channels
- HyperForm: Distance between objects, section plotting, draw bead display available
- For a detailed listing of the all the enhancements in Engineering Solutions click [here](#)

Solver Interfaces

- Many enhancements and bugfixes for each solver interface is made in this release. A short mention of items for each solver interface is given below. Refer to the complete release notes of the [Solver Interfacing](#) for more information.
- Abaqus: Support of erosion or failed elements and Smoothed Particle Hydrodynamic (SPH)
- OptiStruct/Nastran: CSHEAR and CBEAM in Op2/XDB
- RADIOSS: Support of RADIOSS solver Element System, and Femzip-Radioss

General Enhancements

- Linux & Mac GUI:
 - File open and save dialogs are extended to be similar to what's seen on Windows with options for saving favorites, recently visited, and creating folders from the dialog.
 - Fonts are made consistent throughout the GUI, and will reflect closer to what the intended font size versus a fixed pixel size. The fonts scale up appropriately on high resolution monitors.



- Performance with large MBD models
 - Significant work is done to improve the performance of large MBD models that contain flexible bodies. Data loading, contouring, and animations that used to take hours on models that have a high number of modes captured now happens in minutes.
- Animated GIF files can be exported with a transparent background
- Deformable section cuts for linear static and modal animations are supported. The section cut at the frame of reference is followed through the animation as the model deforms.
- Named views created by the user can be saved into an external file and later imported into another session
- Derived loadcases have the file names listed next to the simulation steps to help identify the source of the simulation. This is useful when multiple files provide loadcases that are referenced in a derived loadcase like Steps, Linear Superposition, or Envelope. Switch to the Loadcase view on the browser, and make sure to check the file option under Browser Configuration dialog.
- Sets panel is accessible now from the Visualization Toolbar

- Build plots panel can be used to plot MBD models containing flexible bodies
- All graphic manipulators used in Section Cuts, Image Planes, CFD rakes provide visual feedback to the amount of translation and rotation in displaying the values as they are modified.
- A couple of preference statements allow users to create section cuts with the clipping plane option turned on, and the normal reversed by default.
- Entity highlighting color matches the foreground color that gives better visibility against white background settings.
- In the linear animation mode, the number of frames to deform the model from initial position to final position is increased to 25 frames (from 4). The animation looks smooth with increased frames, while the users can control this number via a preference statement if desired.
- An option under Visualization Preferences allows the user to change the rotation behavior in the graphics area, if any specific rotation center is defined. When this option is active the model rotates around the center of all current visible entities, whereas the model rotates around the center of the screen when the option is inactive. The different behaviors can only be seen, when the model is not positioned in the center of the screen.

Resolved Issues

- Resolved the issue of slider position for lighting direction getting incorrectly restored after closing the Preferences > Options dialog.
- Application crash prevented in an unsupported use case of mixing nodes on a section and regular nodes of the model for a node path measure.
- Selection of all entities using the extended entity selection menu failed if a previous operation of "by window" or "by plane" or "by id" was performed. This issue is resolved.
- Save and Retrieve for extended selection of any collectors works as intended.
- Extended entity selection of nodes By Plane caused application error when a user reset the selection and tried to add further. The error is resolved.
- Extended entity selection mode of Subtract considers correctly the visibility setting of all or displayed.
- Selection "by contour" is updated for corner bound results such that the criteria is compared against all values even if some of them are not available. Previously the search stopped when a first corner value of N/A was encountered.
- A logic error in the measure drawing code that calculates measure visibility is corrected. In some cases, the visibility of each measure in the group that consists of multiple items was incorrectly calculated.
- Missing subcases from HvTrans when converting an Op2 file is resolved after a fix to the Op2 reader. In these cases, the reader was erroneously sending a subcase ID of 0 for some subcases.
- Issue with HvTrans that was producing an error in retaining the component names is resolved. Part ID references were re-indexed in the translation causing the name mismatch.
- Element sets are correctly exported into the H3D file from a solver input deck when translated in HvTrans.
- A first time section of the type N1, N2, N3 was getting converted to Y-axis as there was a base node reset happening. The issue is resolved.
- Section cut drawing code corrected that was hiding System plot, Vector, and Tensor plot on solid element models.
- Section cuts with a chosen base point were not created correctly if a tracking system is active on the model. The tracking transform is considered to the section's base node to correctly position the section.
- An issue with the slider resolution of section cut panel resulted in unintended changes to the cut locations when switching windows containing models of different dimensions. After modifying a

section cut in one window, if a user switches to another window that has sections as well, the location of the cut is modified. This issue is resolved.

- Older version of NVidia driver such as 319.17 showed performance problems with tensor drawing. The drawing code for tensors is updated to circumvent the performance issue on these older drivers.
- Some specific ATI cards showed issues with drawing measure text when a section cut is active. The drawing code is updated to account for these specific issues.
- Solids having duplicates on the interior faces caused display issues as not correctly shown. The drawing code is updated to account for duplicate solids.
- H3D export of notes having special character of (double quotes) "" or square brackets caused problems in restoring back the text. Measures in the model could also be not restored properly because of these special characters. Added escape character handling for double quotes and square brackets so the text is parsed correctly.
- In a continuous improvement of display performance, the speed of managing visibility of sets is increased with large models having a high number of entity sets.
- Context menus configured in Preference files are correctly loaded when the scripts refer to file path with spaces.
- Custom preference files registered and stored as a relative path to the user home had problems restoring and will not be seen on start up. The initialization code is adjusted to account for relative paths and the issue no longer exists.
- Session files correctly restores the settings of legend, font size that were previously saved by the user.
- Legend file saving and restoring considers all the settings saved by the user including Log interpolation, font size, style, and so on. There was an issue previously with GUI not updated correctly when loading a legend file.
- Deformed shape settings could not be applied with Apply Style in a few cases where Undeformed view is shown as well. This issue is resolved.
- MBD models that do not have any graphic entities had display issues with markers, either the markers were too small or not seen on model read. The issue is resolved.
- An incorrect clipping of the model resulted in some parts disappearing during animation. This issue was seen with large unit models that have small components, and the display of all parts are turned off except the small dimension part – as in the case of an MBD simulation of a car navigating a long winded road. The drawing is corrected so the parts don't disappear during animation.
- Airbag particle results from an LS-DYNA simulation created an invalid state when derived results were created using Result Math. The issue is resolved.
- PowerPoint publishing initiated application errors with H3D export when window is hidden from the active page due to another window in expanded state. The error no longer appears with a fix in the export code to address this use case.
- Improved support of PowerPoint 2013 master templates in publishing.
- If the same MBD model with Added Objects is loaded in multiple windows, deleting one window Added Objects caused a deletion of the same in other windows. This issue is resolved.
- Measures referencing entity (nodes, elements) with IDs beyond the precision of a single float returned a truncated number in query and curve plots. The precision is changed to double for large numbered entities in the get and set functions of the API.
- Apply Style on measures considers the "Use tracking" option correctly when applying the setting to measures in other windows or pages.
- Session files and Report Templates reading caused some measure based live linked curves to not animate correctly. This issue is resolved.
- Application crash when creating measures by node path is prevented in some cases where the IDs specified are invalid.

- Apply Style did not apply the settings on a page if one of the windows is in an expanded state. The issue is resolved by considering the hidden windows as well.
- Query on vector results for components returned as N/A. This issue is corrected.
- Deform scale factors were changing after element selection from a collector in some cases. The selection process was resetting the simulation step causing an un-scale. This issue is resolved.
- H3D files are loaded successfully on Windows machines by a drag and drop operation of the file on the application shortcut.
- A report template saved with HyperMesh as the active client of the session did not restore the time scales which resulted in HyperView windows not animating correctly. This issue is resolved.
- BeginInterpreter block in a report template that runs user defined tcl commands was getting executed only on the first instance of a report template. Subsequent overlay or appends did not execute the tcl statements. This issue is resolved.
- Last accessed directory is made consistent across all clients of HyperWorks Desktop as follows
 - If the preference statement *DefaultFileAccessFromWorkingDirectory is defined and set to true, the initial directory is set to the start-in directory, and the behavior is to always save the "last accessed directory".
 - If the preference statement *DefaultFileAccessFromWorkingDirectory is not defined, the initial directory is set to the value saved in the hwsettings.xml file, and the behavior is to always save the "last accessed directory".
 - If the preference statement *DefaultFileAccessFromWorkingDirectory is defined and set to false, the initial directory is set to the value saved in the hwsettings.xml file, and the behavior is to always save the "last accessed directory".
- Window synchronization toggle is correctly updated when a new page is added or a report template is instantiated that brings in additional pages containing multiple windows. The toggle status was not updated previously misleading the user to think the state is turned on.
- Coordinate systems size is adjusted to be drawn more consistently between models of different dimensions.
- The Next, Previous, First and Last button on the Animation minibar were not functioning properly with Derived loadcases in some situations. This issue is resolved.
- Application crash resolved when exporting plot styles that had settings captured for Projection rule.
- Note attachment to elements as a default option with preference statements caused some issues when creating a note the first time, in a few cases resulting in application crash. In a few other cases, the H3D was not usable. These issues are resolved.
- H3D export of models with element erosion is handled properly and only the displayed elements are exported if chosen by the user, instead of a full model, thus reducing the file size.
- In some cases of defining measure on nodes of a section cut, an H3D exported file did not read in properly. The reading code is updated to handle these special cases.
- Result math on 1D element results works as intended.
- Circular symmetry combined with a planar reflection in that order is corrected to show the correct visualization of the operation.
- Vector drawing is corrected when a tracking system is active. The values have been correct, only the drawing of vectors did not consider the tracking transformations.
- The display state of the vector plot with hidden legends or arrows are stored and restored correctly in session and report templates.
- Plot styles capture the colors for vector directions correctly, so the save and restore work as intended.
- The section cut width in HyperView Player matches as that's seen in HyperView.
- When the "use part color" option under preferences is checked, the model display now considers any attributes for displaying the material styles on the part.

- Saved view with highest ID was activated if the user had auto-save turned on by having *SetAutoSaveInterval() in their preference file. This resulted in the view getting updated after the elapsed time interval for saving. This issue is resolved and the view is unaffected.
- Mesh line display status is correctly applied from an Advanced Query dialog.
- Browser tree is updated to show the display status of components after an Unmask operation.
- File open dialog appears on the same monitor that HVP is running for a case with multiple monitors.
- Fonts and colors for node sets were getting modified unintentionally after some edits on the set contents. The issue is resolved.
- Updated the Notes panel code to show the field names drop down list when elements are added using the extended selection or multiple select option is selected. Previously adding items using this dialog did not result in the drop down being refreshed.
- Escape key takes the user out of Selector, Show/Hide/Isolate modes.
- Expression builder dialog for result math allows creation of user created datatypes even when the checkbox "Display alias" is turned off.
- While in the Contour panel if the user creates a coordinate system, the panel is refreshed so the systems can be used for any transformation of the results. Previously the users had to switch panels to refresh the resolved-in systems list.
- Turning off layers coming from report templates also turns off the notes belonging to that layer.
- Systems that are turned off get turned on with a copy window and paste operation. The visibility is correctly captured now in the copy paste operation.
- On Linux platforms only, if a screen or animation capture was tried to saved to a location where the write permissions do not exist, the status erroneously showed as the file saved successfully. A message appears now that the file cannot be saved to the location due to permission issues.
- Set visibility is correctly captured and restored in H3D files.
- Accelerated graphics produces a good representation of quad elements in the model. No artificial triangles are seen in display.

12.0.115 Release Notes

New Features and Enhancements

- A new option for tracking systems to align with global coordinate system is provided to transform the outputs back to global directions. This functionality is useful in obtaining intrusion outputs in the global orientation while removing the rigid body motion. Displacement based Contour and Vector plots, as well as all location based Measures like Position, Relative Displacement, Distance Between are affected when "Use tracking" option is selected.
- Deformed shape of the model can be exported taking into effect the active tracking system. It is quite helpful in evaluating clearances and part interactions due to deformation in packaging studies. The deformed shape resembles exactly as seen in the graphics window with tracking. Align with global option on the tracking system influences in which system the coordinates are output.
- Query output of nodal positions is supported when tracking system is active. Align with global option on the tracking system influences in which system the coordinates are output.

Resolved Issues

- Recent version of NVIDIA drivers caused an error message "error C6022: No input primitive type" and failed to load Accelerated Graphics on startup. This issue is resolved.

- Drawing code updated so that FLC/FLD plots appear properly on machines with ATI graphics cards.

12.0.112 Release Notes

Abaqus Interface

- The Abaqus 6.13 reader has been added for Windows 64bit and Linux 64bit. HyperWorks 12.0.112 also supports these versions: 6.10, 6.10-EF, 6.11, 6.12.

LS-DYNA Interface

- FEMZIP reader upgraded to FEMZIP 6.05 libraries to fix issues reported in FEMZIP 6.0 libraries.

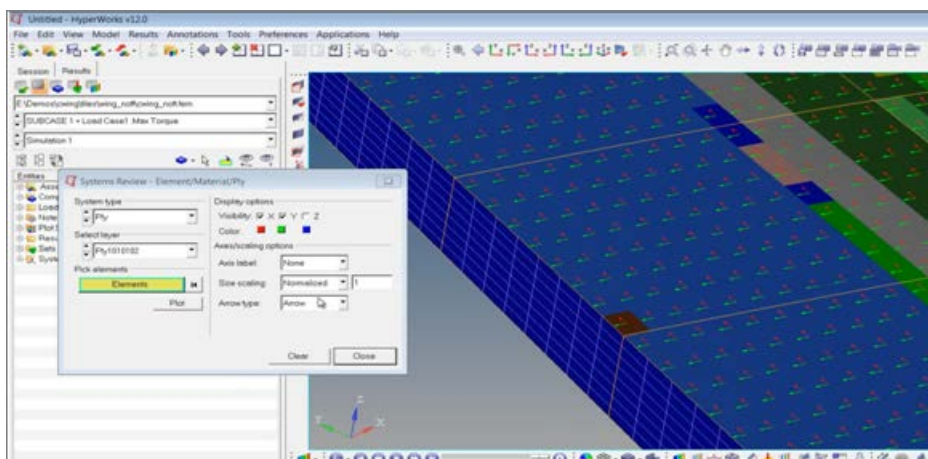
Resolved Issues

- Math operations on 1D element results enabled for all solvers.
- Magnitude for Incremental Distance is restored to a scalar difference of change in distance as was the case in v12.0 and earlier.
- Ansys Interface
 - Issue with results for the solution with large number of simulations where vonMises results for the last simulation were not displayed has been fixed now.
- AcuSolve Interface
 - Selecting “Velocity_magnitude” as a result type for contouring will use volume strain rate for contouring.

12.0.110 Release Notes

Systems Review

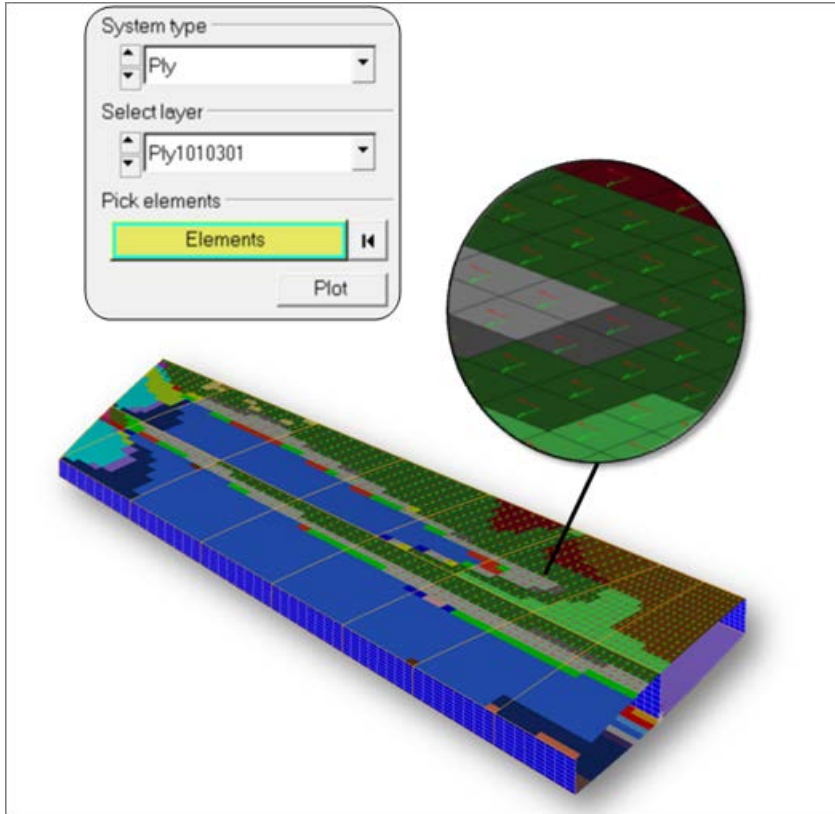
The Systems Review dialog allows you to review the orientations of various element systems (1D, 2D, or 3D), as well as material and ply systems. Reviewing these systems enable engineers to assess the integrity of model and understand the results in significance to the system of interest. Element systems can be reviewed to check normal orientations, proper connectivity, or the beam orientations. The system plots can be overlaid with Vector and Tensor plots to confirm the results and their orientations are making sense. Different display options are available that customize the appearance of the system plot.



Composites Post-Processing

- **Material and ply system review**

Material and Ply systems can be plotted using the Systems Review dialog for the composite models. Advanced Result Math template must be selected in the Load Model panel when initially loading the model and result file. In addition, a solver input deck is also required as the model file. The systems review can be overlaid with tensor plot for verification.

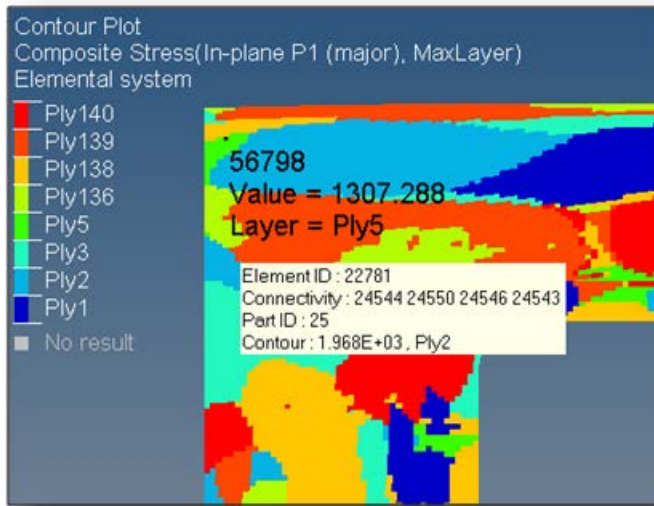


- **Tensor plot of composite stresses and strains**

Composite stresses and strains can be tensor plotted in elemental, material, global or user defined systems. The transformed tensors can be exported to a .csv file for any external processing or hand calculations.

- **Min/max ply identification by name**

MaxLayer, MinLayer, and ExtremeLayer plots in the contour panel that identifies the min/max/extreme values across plies are enhanced to display the names for easy identification of significant plies. In prior versions, the legend only showed the internal ID of plies that had maximum/minimum/extreme value of results across all the plies. A user had to use the Quick query function or the Query panel if he wanted to know the actual ply name. Now the ply names are shown directly in the contour legend with color coding of individual or a group of plies. Measures, Query, and Notes defined for these plots produce the ply name that has the min/max/extreme value, as well as the corresponding actual numeric data of the result.



- **Ply count**

A new aggregation mode called as Count provides a count of layers associated with each element, node. Used in conjunction with the Value filter, users can determine the number of plies failing a certain value criteria.

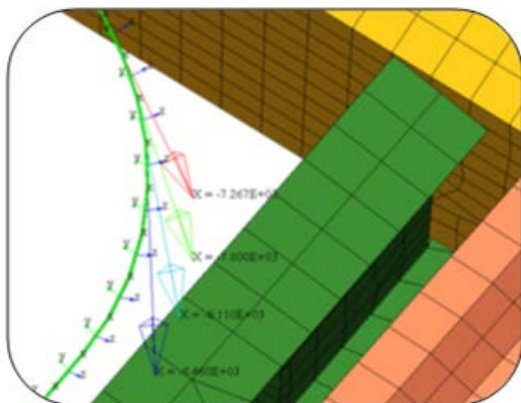
1D Element Post-Processing

- **Element orientation review**

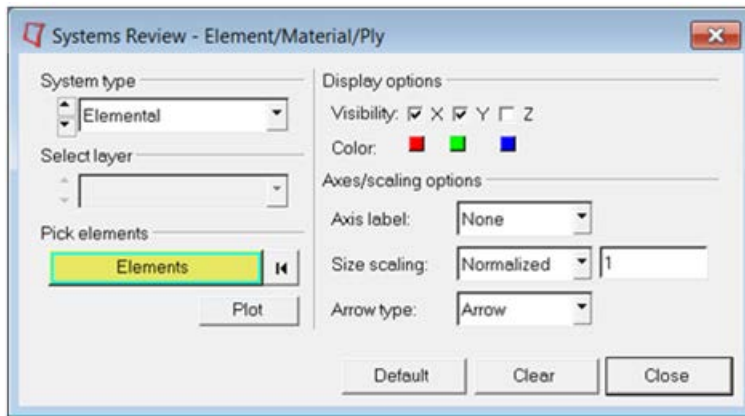
1D element systems can be reviewed from Systems Review dialog. Beams, bars and bushes modeled as 1D elements can be reviewed for validation of model, and analysis of direction of forces and moments. Different plotting actions for displaying the element system are available. 1D element system display can be overlaid with vector. Element system review supported for OptiStruct and Nastran solvers.

- **Forces and moments vector plots**

1D element forces and moments can be vector plotted. The results are shown as vectors with the functionality available to transform the forces and moments to any coordinate system –from elemental system to global or to any user defined system. These transformed forces/moments can be exported to a .csv file for any external processing or hand calculations. Element orientations and vector plots can exist simultaneously. 1D element forces and moments are supported for OptiStruct and Nastran solvers.



Overlay of 1D element system with element forces

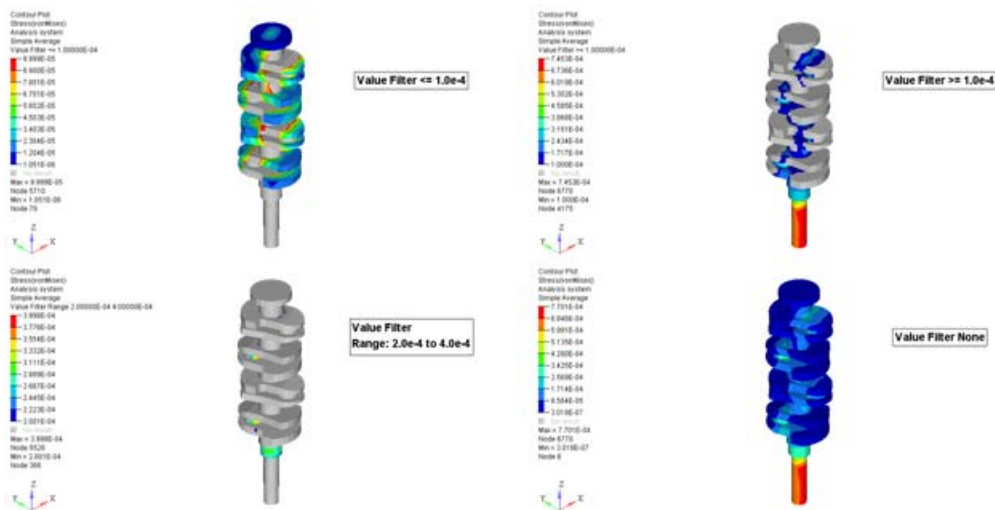


- **Shear and axial resultants**

Vector plotting is updated to provide shear and axial resultant of individual components. Typically relevant for 1D element force/moment vector plots, the shear and axial resultants help calculate the failure limits of fasteners and joints. Users can select anywhere from a single component to all the resultant combinations in one plot with values displayed at the tip of the arrows. The display options for Vector plots are now placed into logical tabs (Plot, Display, Legend, Section).

Value Filter

The Value filter available in the Contour panel allows specifying threshold criteria for processing results only in a specific range. The Value filter discards any values not meeting the specified criteria (greater than or equal to (\geq), less than or equal to (\leq), or a Range). It can be a useful tool for isolating the 'hotspot' areas in a quick manner. Value filter along with the Count aggregation mode allows users to count the number of layers meeting a certain criteria (for example, counting the number of plies exceeding a failure index).



HyperView Player

- Section cuts can be created interactively in the Player from the Quick Section Toolbar and context menu. Section cuts with normal directions of X, Y, Z can be easily created, or any arbitrary plane orientation by picking 3 nodes. The onscreen manipulator allows users to position the section at the desired location or traverse the cut portion of the model. Reversing the plane direction, or setting the transparency on the cut portion, along with displaying the clipping plane or not are available from the context menu.
- Progress bar is shown when loading files or results and animations. This is useful when reading large models, giving the user feedback on the progress of the number of steps of animation loaded.

- Model Title note can be turned on/off from the context menu.
-

General Enhancements

- Performance of processing results from an Envelope loadcase spanning over hundreds of loadcases is improved significantly.
- Performance improved for Browser operations of displaying parts with show/hide/isolate on large models.
- Models can be animated or deformed based on any nodal vector, including user created nodal vectors from Result Math. Deformed Panel allows the users to pick a result type to deform and animate.
- A new extended selection method of "By Face" is supported for element and node collectors to select entities on continuous surfaces bound within a feature angle. The feature angle set under Preferences > Options > Visualization affects the entities on the face.
- A new extended selection method of "By Plane" is supported for element and node collectors to select entities lying on a plane. Users can set the tolerance in the plane normal direction to select the entities.
- The export options for H3D are exposed in a single dialog where the file name is specified. The same options are available under Preferences > Options as well.
- Entity sets of nodes, elements, parts created in an active session can be excluded from H3D export. By default all the sets are written to the h3d file. To reduce the file size, users have the option now in the Export H3D Options dialog.
- Assembly Hierarchy allows parts existing in multiple assemblies similar to HyperMesh structure. The hierarchy will be read from a solver input deck created in HyperMesh, loaded as the model file. An environment variable (HV_ASSEM_DUPLICATE_PARTS) should be enabled.
- A new preference statement *DefaultMaxFrameRate allows setting the default maximum frame rate animation on start up. A value between 1 and 100 is required to be set. This statement can also be added to any custom user preference file and will be observed.
- Commands for getting status of display area toolbar, command window, and panels have been added. These commands are useful in customization scripts to maximize the display area and knowing the status of the GUI
- The generaci ascii file format (*.hwascii) is supported for MAC.
- Node selection by ids can be specified when aligning a video with a model for the 3D image planes. This is available for both 2-point and 3-point alignment methods by clicking on N1, N2, or N3 collectors.
- ApplyStyle functionality supports User defined systems. The systems created in one window can be copied to other windows by executing the ApplyStyle context menu.

Resolved Issues

- Erosion models that exhibit element failures in transient animations are exported correctly into the H3D. The resulting file size is also smaller exporting only the displayed elements on the screen.
- H3D export bug corrected so that all hidden components are not available after exporting into h3d if the "Include masked elements" option is not active.
- "Include masked elements" option for H3D export also considered for publishing a session (html or ppt) that has animations captured in H3D format.
- When working with two PowerPoint files that have embedded HyperView player, double clicking to open the h3d resulted in an incorrect file loaded. Unique names for the temporary files created resolves the issue and users do not see the issue anymore.
- On MAC machines, the Symmetry with respect to markers on an MBD model are restored correctly in H3D

- Implemented saving and restoring of draw style and draw size for entity sets written out from HyperView.
- Application crash is prevented in cases a tracking system is deleted from the browser.
- Unexpected hang of application resolved when resetting Synchronize animation (from Tools menu) using the Video Overlay feature on Linux
- HyperView window gets locked after certain copy/paste window operations from the session browser. Subsequent operations resulted in application errors. This issue is resolved.
- Measures defined between overlaid models are listed in the browser just as the Measure panel
- Measures and Note transparency and autohide settings are captured in H3D export to see them as saved in HyperView Player.
- Measure expressions are correctly offset with page and group ids when used in Report Templates with overlay or append mode. Live linked curves pull the data from appropriate measures in these situations.
- Cascading menu items on Linux machines are refreshed correctly when moving between different items.
- Symmetry dialog displays any previously selected parts on re-opening it for users information.
- Invoking symmetry changes affected the note attachment type from a coordinate to node. This issue is resolved
- Preference file statement for setting the note fonts, *NoteFont, is considered for notes created using "Multi select" operation as well.
- Font style correctly captured and restored in H3D with node path measures.
- Image capture on Windows XP-32 updated to not include the dialog remnant for captured images.
- After reading legacy session files (10.0 or older), the complex results filter was incorrectly set on a new session file save. This issue is resolved.
- Deformed scale factor was incorrectly reset to 1.0 when loading a session file and saving it again without visiting all the pages. This issue is resolved and all the scale factors are saved as they are defined in the original file irrespective of switching any pages.
- Legend formatting is not applied under Linux when a session file is loaded with fonts that do not exist. A fix is implemented to default the font types only and not affect other legend attributes.
- Changing the dof's for a tracking system inadvertently applied a contour if it had been cleared previously. Users are forced to clear the contour again. Checks are implemented to see if a contour plot exists or not, and updated the tracking system properties does not create a contour plot automatically.
- Undeformed shape color is captured and restored correctly in window copy/paste operations.
- Fixed an issue where the file open dialog of the Sets import/export GUI invokes in the background on LINUX only.
- No results in hwascii format as shown as N/A instead of zeros.
- Node trace is available on reflected parts of symmetry.
- TCL plugins (result math operations) on hex 20 noded elements with complex results resulted in memory overflow and crash of the application. This issue is resolved.
- TCL plugins (result math operations) support complex data also. Results of nodes/elements can be extracted for a complex loadcase and math operations scripted in the plugin.
- Result math operations can extract yield stress or material ID from the solver input deck for materials bound to parts as in the Radioss or LS-Dyna solvers.
- Rake outline visibility is captured and restored correctly in H3D Export.
- Quick window selection did not work correctly when tracking system is present, wrong entities were selected. The tracking system transforms are appropriately dealt now while processing a quick window selection of entities.

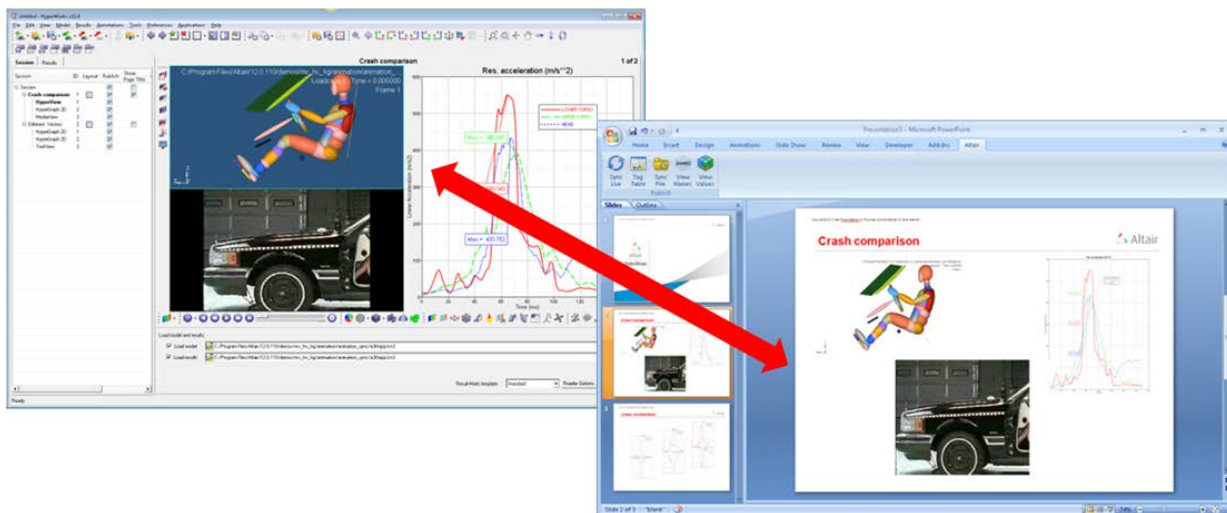
- The HVP plugin registration script, regHVPDll.bat, is updated with proper version of the plugin from 11.0 to 12.0.
- Session file does not get saved when connection to license server is lost even after choosing yes to do so. The issue corrected to account for forced shutdown of the application, and save on exit on user option. When HyperMesh is the active client at the time of lost license connection, the app crashed on saving. This is resolved as well.
- Command window closes with "exit" command and could not be opened any more. This issue is fixed.
- Quick query selection sometimes caused the last entity selection to be used for a next operation like contour plots. The selection interference with panel collectors is avoided.
- Sessions with multiple windows each having a model at different step increment wasn't restored correctly when a Derived Loadcase existed. The resetting of the simulation index to the first one is resolved.
- Application errors showed up while dragging the current time slider in Synchronize Window Animation dialog. Issue is resolved.
- Node path algorithm for Measures updated to use the shortest distance between selected nodes.
- A regression introduced in v12.0 of HyperView is resolved where the thickness of border of notes and the leader lines are different.
- The magnitude calculation for Incremental Distance is corrected to vector addition rather than a straight difference of magnitudes of two vector positions.
- A better algorithm for Iso-surface is implemented that handles solids having one or more "No Result" nodes.
- When reading partial input decks that have grid and element definitions in multiple include files, the model renumbering was happening subsequently causing wrong results to show. This issue is resolved by updating the model correctly accounting for removed elements that had invalid connectivity.
- HyperView Player on Linux crashed while loading very old h3d files (created v7.0 or prior). The legacy reader is updated to handle this case.
- Minor update to the H3D Export Settings dialog to accept integers and floating point numbers for the compression factor. Application error or no compression resulted previously using integers or blank in the entry field.
- Application crash prevented when using Preference statement *NoteAttachment("Node") for note attachment.
- GetValueList API on measures crashed the application when querying entities that do not have any results. This issue is resolved now and should help automation scripts that unknowingly query for the entities with N/A values.
- Users can launch "Optistruct" and "Radioss" solvers through "Applications è OptiStruct/Radioss" on Linux and Mac machines which wasn't possible before due to an issue in the invoke script.
- When evaluating Envelope derived loadcase on files generated from Radioss, LS-DYNA, or PamCrash natively or converted to H3D, the results for the aggregation were incorrect in certain cases. Checks have been added to handle these scenarios.
- Note AutoHide setting was not respected in image screenshot or PowerPoint export. The logic in the rendering code is updated to consider the setting when capturing images.

HyperGraph 13.0 Release Notes

13.0 Release Notes

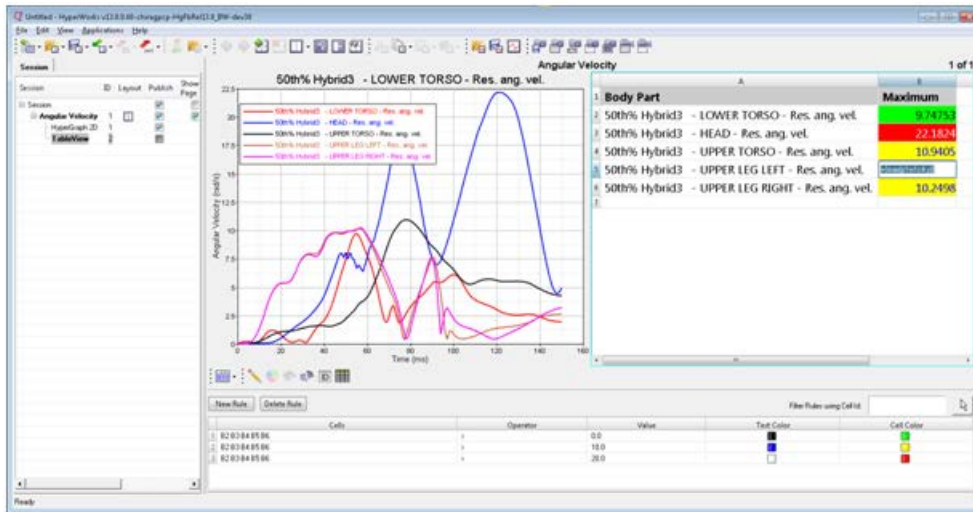
New Publishing Features

The new HyperWorks Desktop (HWD) PowerPoint Publishing feature helps to reduce the work and effort in generating and updating Microsoft PowerPoint (PPT) presentations from HWD sessions. This goal could be achieved by establishing a live link between HWD and PPT. The synchronization could be triggered either from HWD or PPT using a PPT Plug-in. Changes made in the PowerPoint PPT presentation will be preserved when new runs will be post processed after updating your session; for example, using Report Templates. You can predefine the PowerPoint layout for the first publishing using PowerPoint Master Templates. The visualization of broken connections in PPT allows easy debugging if the PPT presentation will not be updated correctly. All HWD export formats are supported: Anim, GIF, AVI, H3D, PNG, JPEG, BMP, and TIF. Dynamic linking of HyperGraph notes with PPT allows you to update text fields with new results. Live links are supported on Windows, Linux will be supported by data exchange over the file system using the PPT Plug-in.



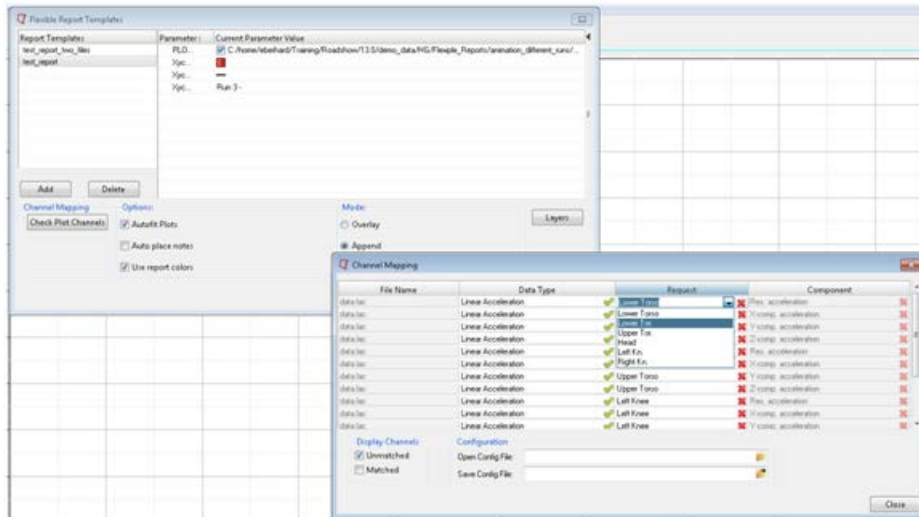
TableView

The new TableView feature creates Excel-like spreadsheets inside of HyperWorks Desktop. Synchronization with HyperGraph can be done using Templex expression inside of cells and allows you to create summary tables which update accordingly using report templates. Value-dependent formatting allows the easy detection of critical KPI's. Basic functionality to format and edit tables is available.



Flexible Report Templates

The new easy-to-use Flexible Report panel replaces the Report Panel from earlier HyperWorks Desktop versions. It makes reports more flexible by giving the user advanced options to handle plotting channels during the import. All channels referring to a file could be skipped, their types, requests and components could be checked and reconnected in case there is a mismatch. The reconnected channels can be saved and reused using configuration files. More user-friendly color and style settings are possible for parameters created with the Parameter browser.



Marker Tracking

The Marker Tracking feature in MediaView allows you to track any kind of markers or objects in MediaView as the video is animated. This provides numerical data which can be displayed and animated in the HyperGraph. Manipulation of the tracking data by applying units or other math operations allows the comparison of tests and simulations. Markers can be added via mouse or API. Graphical options like show/hide tracking lines and marker IDs or show tracking markers and traces only makes it easy for the user to analyze the testing video.



Engineering Solutions

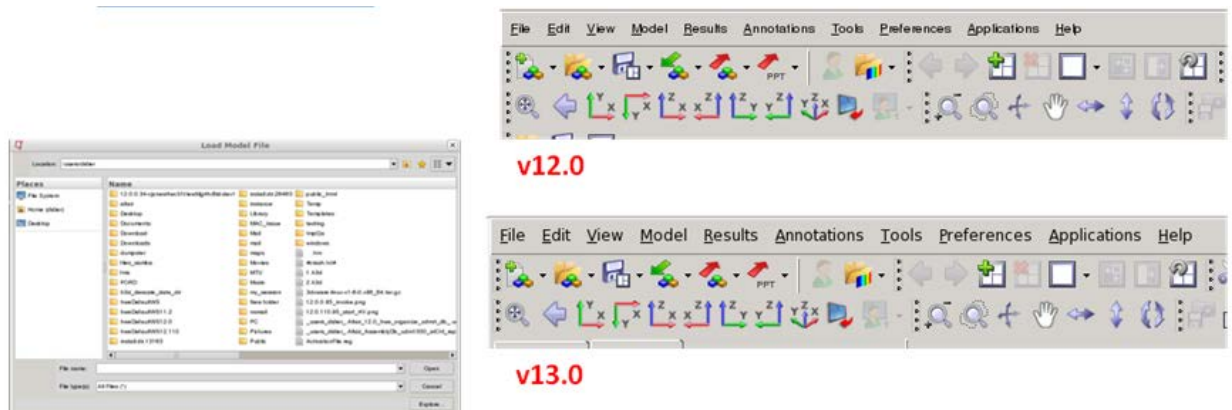
- NVH: Model and Test correlation tool MAC and CoMAC is added
- Crash: Diagnostic and model compare tools new, global control of data from browser, and flexible report templates for mapping channels
- HyperForm: Distance between objects, section plotting, draw bead display available
- For a detailed listing of the all the enhancements in Engineering Solutions, [click here](#)

Solver Interfaces

- Many enhancements and bugfixes for each solver interface is made in this release. Refer to the complete release notes of the [Solver Interfacing](#) for more information.
- Abaqus: Support of erosion or failed elements and Smoothed Particle Hydrodynamic (SPH)
- OptiStruct/Nastran: CSHEAR and CBEAM in Op2/XDB
- RADIOSS: Support of RADIOSS solver Element System, and Femzip-Radioss

General Enhancements

- Linux & Mac GUI:
 - File open and save dialogs are extended to be similar to what's seen on Windows with options for saving favorites, recently visited, and creating folders from the dialog.
 - Fonts are made consistent throughout the GUI, and will reflect closer to what the intended font size versus a fixed pixel size. The fonts scale up appropriately on high resolution monitors.



- Performing math operations on curves is now possible during curve import using the **Advanced** button in the Build Plots panel. U and V vectors can be used in the X and Y vectors to perform any kind of math operations during import.
- Unit Scaling on/off checkbox is now provided in the Options panel.
- Animated GIF can now be exported with a transparent background
- HGTrans in batch mode supports now an option to extract all time history curves from Abaqus .odb files.
- Evaluation of Templex Expressions in 3D plot curve labels is now available.
- Update of the ELOUT reader to read strain output from LS-DYNA ELOUT file.
- Update of the hgrpc3 reader to support Simpack file format.
- Support of Templex expressions like {v.HWDatatype} - {v.HWRequest} - {v.HWComponent} which refer to u and v vectors from type file like already available for x and y vectors.
- Curve labels prefixes and suffixes will be displayed in the Plot browser as well as the property area.
- Both curve label prefixes from **Multiple Files** and from **Advanced Options** can be used using the **Multiple Files** feature. The prefix from multiple files will be displayed first.
- New reader and file filters in the Build Plots panel are added for the Tiem Orbit property files that are used by Adams and MotionSolve is now available. The supported files are from type are .bus, .reb, .dpr, .spr and .bum. A file filter for .plt files was also added.

Resolved Issues

- Copy & paste single/multiple columns of the coordinate info value into Excel works now again as it did in HyperGraph 10.0.
- Templex functions "saefilt95" will handle the beginning and the end of the filtered curves according to the given number of padding sample in the forth argument, the documentation is updated accordingly. If the number of samples exceeds the limit an error message will be raised.
- Loading bar charts with report templates sometimes took very long and the bar charts looked disarranged. This issue is solved.
- HyperGraph can now evaluate the expression of registered HyperMath functions with capital letters in the HyperMath function name.
- Loading T01 files into failed with an error message while the readers processed the velocity components when no time vector was available. This issue is solved.
- Selecting files in the left window of the **Multiple File** tool used to create an unsorted list in the right window. These files are now listed in dictionary order.
- The legend leader position is updated using the micro dialog menu activated by double-clicking on a HyperGraph legend.
- In the Plot browser, entities like horizontal/vertical axis or frame color the default color in the properties table is no longer null.
- Polar plots could be fitted per frame. If in the Axis panel the check box **Fit All Frames** is unset, it will perform a fit for the current frame (frequency).
- Axis changed to dB20 scaling will be saved correctly in the session file
- The rainflow function would crash if the number of bins argument was set to certain values, such as 128. This issue is fixed.
- Any rigid body results using the LS Dyna rdbout reader which was not available in first block (time = 0) and appear at later time stamp will be skipped from the displayed results in the Build Plots panel.
- Overlaying/Appending a report templates multiple times on Linux via scripting sometimes crashed HyperGraph. This issues is fixed.

- Entry fields in HyperGraph are now accessible after using IIHS tools from Vehicle Safety Tools on Linux.
- HyperGraph now labels the rotation angles as RX/RZ/RZ correctly reading both Adams REQ as well as Motion solve *.plt file.
- The names of curves in complex plots no longer changes using the "View Dependency" feature
- The order of multiple curves created with HyperGraph macros by selecting multiple curves via double-click in the name field of the macro panel was different than in the selection.
- The bug that the user is unable to edit text in the text client when clicking the evaluate button more than twice on Linux and Mac OS X machines is fixed.
- One of the great strength of the report functionality is that the user can overlay reports over different instances of appended reports. The bug that the overlay starts always at page 1 is fixed.
- Preference statement *EnableTemplexCurveNames(true/false) works as intended.
- The curve offset in X direction was doubled for some file formats, for example, Madymo. The issue is fixed.
- The scaling of -1 was ignored when the Unit Scaling feature was on. This works now in 13.0.
- The hgthp+.exe reader is now able to read thp files which contain non-ascii characters.
- In certain plots, the y-axis data has being lost opening an existing session file. The issue is fixed.
- When the horizontal axis was set to the logarithmic scaling, the grid line color of vertical axis changed to the same color as the vertical axis label's color. The issue is fixed
- Calculated time vector reading ISO MME files will be displayed without numerical errors.
- When reports templates are overlaid in HyperGraph all notes and curves will belong to the same layer. The error that notes which are not attached to curve will be in a different layer after saving and reopen a session created with report templates is fixed.
- Reader for ELFORCE output in punch format from FRF analysis supports the new element force type 33.
- ABF file export works if curves have the same label. The additional curves that have the same name will be numbered.
- Strain results SXY, SYZ, and SZX from Marc t16 file will be read correctly in HyperGraph. In previous versions these values where wrong by the factor of two.
- Axis labels in HyperGraph Complex Plot are now positioned correctly. Previously the first character of label has always been centered in the mid of the Y-axis.

Known Issues

- No publishing support for the new TableView client
- Templex expressions which refer to HyperView measures on different pages than the TableView client will not be evaluated at a table refresh.

12.0.112 Release Notes

Resolved Issues

- **LS-DYNA interface:** Issue with reading rwhorc data in binout file has been resolved.

12.0.111 Release Notes

Introduction

HyperWorks HW12.0.111-HWDesktop contains new features for supporting frequency, and amplitude dependent bushings. The implementation consists of four different components that work together. Collectively, they provide a comprehensive capability for identifying bushings from test data and, subsequently, using the bushing models in a simulation. The components are summarized below.

- **The Model Identification Tool (MIT):** The MIT manages the workflow associated with fitting experimentally-measured bushing data to analytical bushing models and storing these in ASCII property files. The models in this context are for frequency, amplitude and preload dependent bushings. Two dynamic and three static models are supported. The MIT is launched from HyperGraph.
- **MIT-Fitting Tool:** This tool is invoked by the MIT user interface. It applies sophisticated numerical methods to calculate model coefficients so that deviations between model behavior and known experimental data are minimized. The fitting tool is an extension to HyperGraph.
- **Bushing Modeling in MotionView:** This is a set of panels in MotionView to instantiate and modify frequency, amplitude and preload dependent bushings in a vehicle model.
- **Altair Bushing Model:** The Altair Bushing Model is used with fitted bushing models in a multi-body simulation. It is a solver run-time demand load library (.dll or shared library) that works with MotionSolve. Adaptors for Adams/Solver™, MATLAB™ and SIMPACK™ are also available. The Altair Bushing Model is not part of the 12.0.111 HyperWorks package. It is delivered separately and it works with 12.0.111.

The Model Identification Tool

The MIT is loaded as a preference in HyperGraph. The MIT includes the following seven tabs to support the necessary workflows for fitting bushings:

- **Select Data Tab:** used for specifying experimental data as input.
- **View Data Tab:** used for examining experimental results.
- **Filter Data Tab:** used for selecting a subset of the experimental data to be fitted.
- **Fit Data Tab:** used for fitting data to models for rubber bushing damping or hydromount damping.
- **Analyze Data Tab:** used for generating the behavior of a set of bushing coefficients.
- **Review Data Tab:** used for comparing model behavior to experimental data.
- **Export Data Tab:** used for exporting analytical bushing model coefficients into an output file.

File Input and Output Methods

- MIT reads in bushing experimental data that are provided in the form of Tiem-Orbit data files. These are ASCII files that follow the Tiem-Orbit specification.
- MIT writes out analytical model behavior in the form of output ASCII files. The most important of these are:
 - Output SPD files. Also in Tiem-Orbit format, these contain the results of a virtual test on a virtual bushing. This file is used by the MIT for comparing model performance to measured data.
 - Output GBS file. Also in Tiem-Orbit format, the GBS (General Bushing System) file contains all the data that is required by an MBS solver to simulate a model containing frequency and amplitude dependent bushings.
 - Output LOG file. This is a text file that contains the output of a fitting operation. Details about fitting iterations, optimization iteration history and goodness of fit are included.

Operating Modes

- **Interactive mode:** This is the normal mode of operation. A simple user interface guides the user through the steps of inputting test data for fitting, reviewing the test data, fitting the data, reviewing the fit and exporting the fitted bushing parameters to a "GBS" file. A PDF file containing the fitting summary is automatically exported.
- **Batch mode:** A script, [mit_batch.py](#), is available to run the MIT in a batch mode. Its primary function is to generate one output GBS and/or one SPD file for one bushing. This GBS file may be used for downstream simulation with MotionView/MotionSolve. There are two principal modes of operation.
 - Invoke `mit_batch.py` with a list of input SPD files (`-i` option)
 - Invoke `mit_batch.py` with a directory containing a set of SPD files for a bushing (`-d` option)

For more information on how to use the batch mode, please refer to the Model Identification Tool Users Guide.

Units Support

- Comprehensive Units support including the ability to change input and output Units for the FIT or ANALYZE operation are available.

Supported Bushing Models

The MIT supports three different static models for each direction:

- **Static Spline:** This model represents static nonlinear behavior. This data is obtained by reducing the measured static hysteresis shape into a curve. The MIT does not perform this reduction—the user is expected to do this outside of the MIT.
- **Conceptual Cubic Curve:** This is a set of two cubic curves that can approximate static nonlinear behavior. The curve coefficients are obtained from the static spline and stored in the GBS file.
- **Constant Stiffness:** This is the slope of the static spline at zero deflection.

The MIT supports three different dynamic behavior models for damping: constant damping, the rubber bushing damping model and the hydromount-damping model:

- **Constant Damping Model:** This is a simple model that can be used for preliminary simulations, or when test data is not available in a certain direction. The damping coefficient is a constant and the damping force in the specified direction is the product of the damping coefficient and the deflection velocity in that direction.
- **Rubber Bushing Damping Model:** This model is used to describe non-linear hysteretic systems. Patterned after the well-known Bouc-Wen model of hysteresis, this model is able to capture, in analytical form, a range of hysteretic cycle shapes matching the behavior of a wide class of bushings. This model is capable of modeling the dependency of bushing dynamic stiffness and loss angle on the frequency of the input excitation, the amplitude of the input excitation and the operating condition of the bushing.
- **Hydromount-Damping Model:** This is a mathematical model capable of representing realistic hydromount behavior. It consists of a set of non-linear differential equations that describes the rubber and fluid dynamics associated with the movement of the fluid in the decoupler membrane and in the inertia track of the hydromount. Coupling between the rubber and fluid is also modeled.

An SPD (System Performance Data) is provided as input to the MIT. This is a Tiem-Orbit file that contains all the experimental data, including:

- Dynamic Stiffness and Loss Angle as a function of Frequency, Amplitude and Preload.
- Static measurements in the form of a force vs. deflection curve.

The MIT outputs one or more bushing property files (GBS files) through its Export Data capability. You can use these output GBS files in two ways:

- As an initial guess for design parameters for subsequent fitting.
- As the design parameters for downstream vehicle simulation.

Users Guide and Interactive Help

A comprehensive Users Guide for the MIT is provided with this release. This information is also available through the interactive Help feature in the MIT.

Fitting Tool

The fitting tool is a utility primarily written in Python that fits experimental data to an analytical model. Some of the features in the Fitting Tool include:

- Reading in bushing input files and generating output files summarizing the fit process.
- Fitting experimental data to a selected model using optimization techniques.
- Generating virtual experimental data given a set of bushing model coefficients in a GBS file.

The user does not directly interact with the fitting tool. The MIT provides access to the fitting and analysis functions via the Fit Data Tab and the Analyze Data Tab.

The Altair Bushing Model

These are a set of object libraries that simulate bushing behavior in a tool capable of performing multi-body simulations. The Altair Bushing Model is provided as a neutral .dll that can function with any software that obeys its communication protocol. Extensively tested adapters are available to make the neutral .dll function seamlessly with these products:

- MotionSolve™
- MATLAB™
- Adams™
- SIMPACK™

The simulation model includes the following capabilities:

- Three models for static stiffness:
 - Static Spline.
 - Conceptual Cubic.
 - Constant stiffness.
- Three models for dynamic properties:
 - Constant damping model
 - Rubber bushing Damping.
 - Hydromount Damping.
- A friction model independent of the frequency/amplitude dependence:
 - The LuGre model is used to define the friction parameters.
 - The Dahl model can be obtained simply by setting certain LuGre coefficients to zero.
- A force-coupling model that lets you:
 - Couple bushing forces and torques.

- Apply a cylindrical or spherical coupling method to the computed forces.
- Computing the bushing response at multiple operating conditions at run time
 - The bushing force has two components: (i) A steady state response (F_0) that captures the bushing behavior at an operating condition P , and, (ii) A nonlinear dependent dynamic force (f) that captures the force transience due to vibration about the operating point P .
 - The operating point, P , is obtained by passing the input displacement through a low pass filter. This filter removes the high frequency content and one is left with the underlying low frequency input. This signal is used to calculate the static response, F_0 .
 - The dynamic response is computed by providing the high frequency component of the signal as input to a rubber-damping model.
 - The total force generated by the bushing is the sum of the static force, F_0 , and the transient response, f .
- Local Structural Stiffness:
 - The mount or structure surrounding a bushing may deflect due to the force transmitted by the bushing. Local stiffness models are provided to model this effect.
- Mount Limits:
 - These are bump-stops that model contact in a bushing and are used to limit the range of motion in any direction.
- Displacement Offsets and Scales:
 - Displacement offsets change the origins of the force-deflection curves.
 - Displacement scales expand or contract the force-deflection curves along the deflection axis.
- Preload and Force Scales:
 - Preload offsets the forces or torques at zero displacement.
 - Force scales let you scale the applied force in a direction by a constant.

Bushing Modeling in MotionView

MotionView has been enhanced to contain a new AutoBushFD modeling element. The AutoBushFD is a frequency, amplitude and preload dependent bushing. A set of panels for instantiating an AutoBushFD is now available. When a simulation is to be performed, MotionView exports the AutoBushFD entity to a supported solver such as MotionSolve or ADAMS™.

Altair Bushing Model in Adams/Car™, MATLAB™ and SIMPACK™

Altair offers Adams/Car™, MATLAB™ and SIMPACK™ integration support for simulation models. Please contact your Altair account manager or support@altair.com to learn about this service.

Platform Support

The MIT is available on Windows-7 Professional, Windows-7 Enterprise, and Linux. Both 32- and 64-bit versions of Windows Operating Systems are supported. For Linux, only the 64-bit version is supported. The simulation model is available on the same platforms.

Licensing

The MIT is a utility of HyperGraph and requires that you load HyperGraph prior to launching MIT. The MotionView extensions require MotionView in order to function.

The simulation model is separately licensed and requires six (6) HyperWorks Units in order to function. These Units are leveled against other HyperWorks products that may be running on your machine.

All of these products use the HyperWorks' patented on-demand software licensing system. This centralized licensing model allows multiple users to access HyperWorks products at any time or place. HyperWorks users can run multiple applications for a flat rate, rather than pay for each license in use.

Known Issues

- **Simulation Model: Static equilibrium for friction model**

Static equilibrium with the friction model is not working properly. It does not work for EFFECT = 'STICTION'.

- **Simulation Model: Licensing is not yet implemented**

The simulation can be run without any licensing at present.

- **Fit Tool: Hydromount fitting does not meet the fit metric for all models**

Fitting for Hydromounts is available in this release. Most test datasets are known to fit well. A few test datasets do show poor fitting.

- **MIT: Displaying results for multiple experiments in the Review Tab**

Curves from multiple experiments, when selected, are shown in the same window. This is incorrect. The workaround is to display one experiment at a time.

12.0.110-TMC Release Notes

Resolved Issues

- Common number format for plot axis and Coordinate Info panel: This issue has been resolved. Both now display numbers in the same format. An extra check-box labeled "Sync Coordinate Data with Axis Format" is now provided in the co-ordinate info panel. When the check-box is selected, the data is in sync with the format of corresponding axis.

The screenshot shows the HyperGraph interface. On the left, there are navigation buttons for 'Find point' and 'Next Curve'. The main area displays a table of data points for a curve named '50th% Hybrid3 - LOWER TORSO'. The table has columns for Pt, X, and Y. The Y-axis values are in scientific notation (e.g., 0.00000E+000, 1.12433E-001). On the right, the 'Coordinate Info' panel is visible, showing 'Show X as:' and 'Show Y as:' options, both set to 'Linear'. A checkbox labeled 'Sync Coordinate Data With Axis Format' is checked and highlighted with a red box.

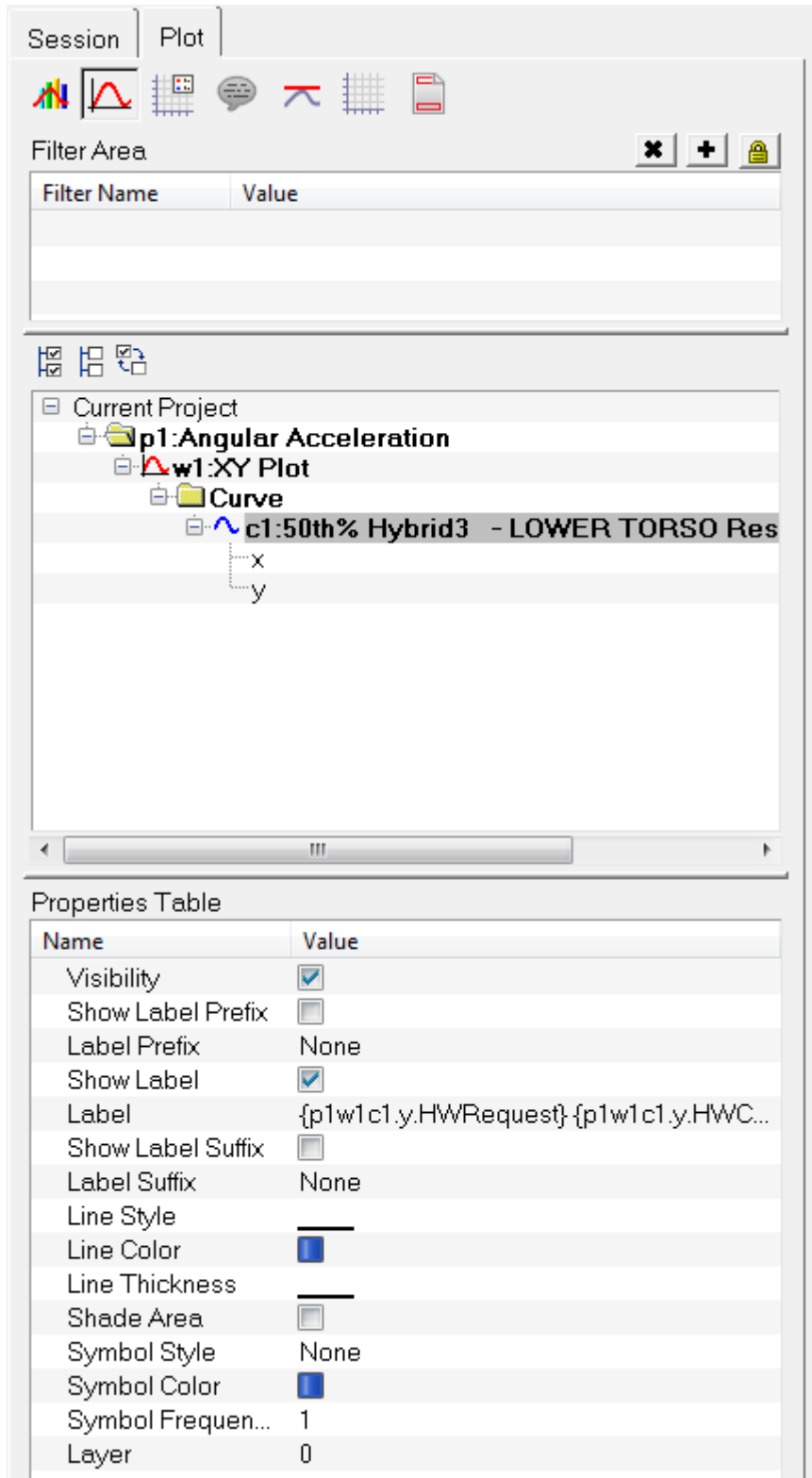
Pt	X	Y
0	0	0.00000E+000
1	1	1.12433E-001
2	2	1.70586E-001
3	3	2.02911E-001
4	4	2.14875E-001
5	5	2.07239E-001
6	6	1.77597E-001

12.0.110 Release Notes

Plot Browser

The **HyperGraph 2D Plot Browser** is located on a tab in a tab area sidebar. It allows you to drive the application and view your session's current plot structure, which is displayed in a tree structure within the browser. You can also perform many of the functions available in the HyperGraph panels through the Plot Browser, allowing you to modify your plot and curve properties in one location. This includes the following:

- Perform Single Curve Math and Multiple Curve Math functions
- Rename curves
- Modify the following curve properties:
 - Curve attributes
 - Axes
 - Plot options
 - Notes
 - Datum lines
 - Plot macros

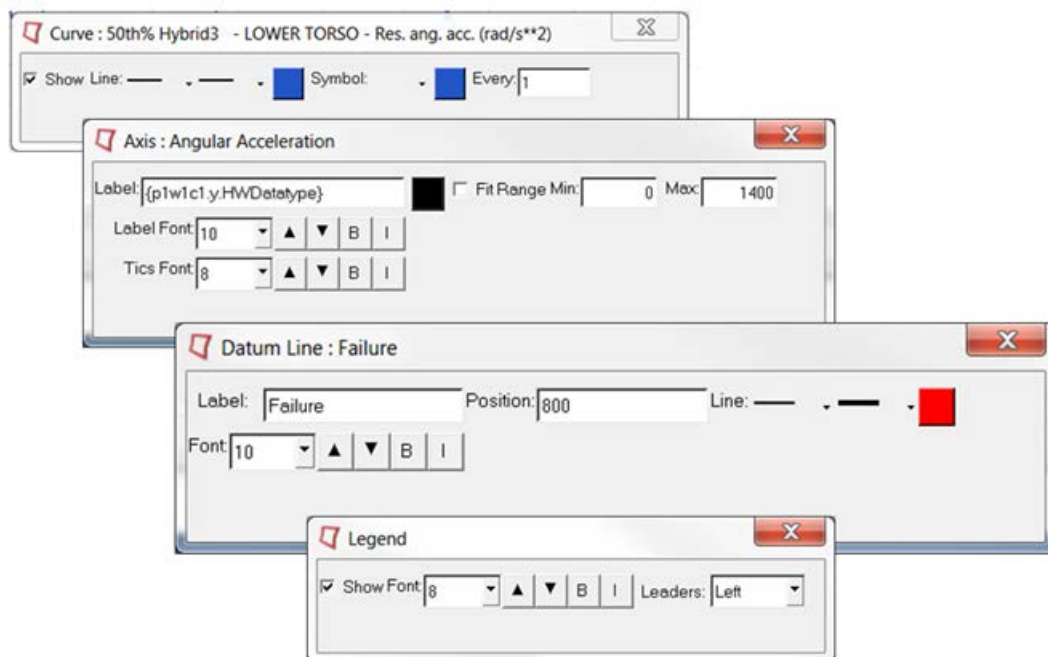


Some of this functionality can be accessed through the Plot Browser's context sensitive menu, while some are accessible through the **Properties Table** in the Plot Browser.

The Plot Browser can be turned on or off using the **View > Browsers > HyperGraph 2D > Plot** menu options. A check mark indicates that the HyperGraph 2D Plot Browser is activated for display in the tab area. The Plot Browser can also be positioned to appear on the right side of the page (the right tab area).

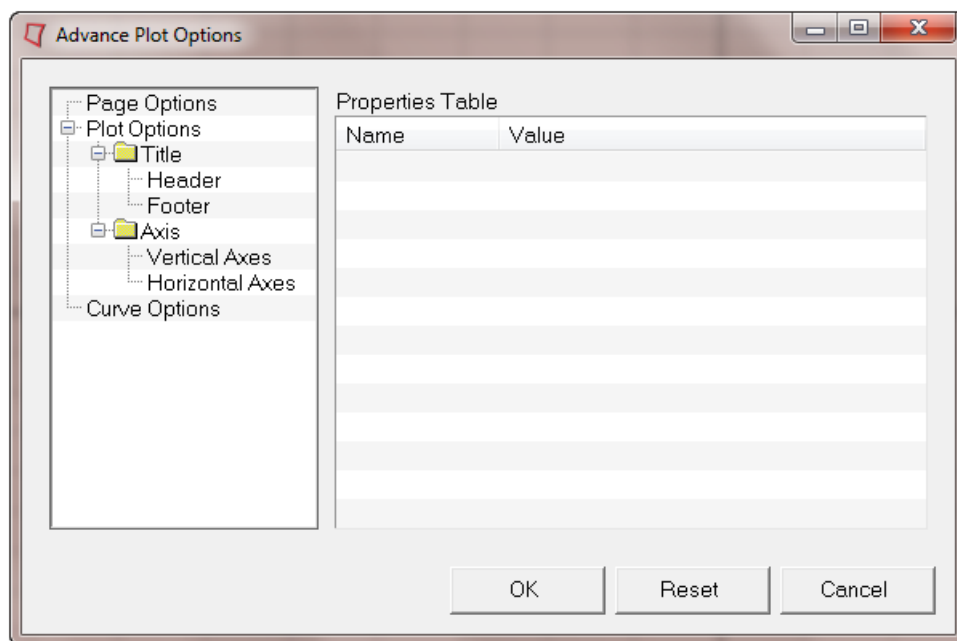
Property Micro Dialogs

In HyperGraph, Property Micro-dialogs contain a subset of the most common functions performed by users. To access a micro-dialog, double-click on a plot "element" to displays the micro-dialog at that location. For example, if you want to modify the header properties, double-click on the header to display the micro-dialog at that location. Micro-dialogs are available only for the XY plot type.



Build Plot Advanced Options

The **Advanced Options** feature on the **Build Plots** panel allows you to apply many options at once to the session during plotting. Curves created in this manner are added to the session in a new layer. From the **Build Plots** panel, click **Adv. Options** to display the **Advanced Plot Options** dialog.



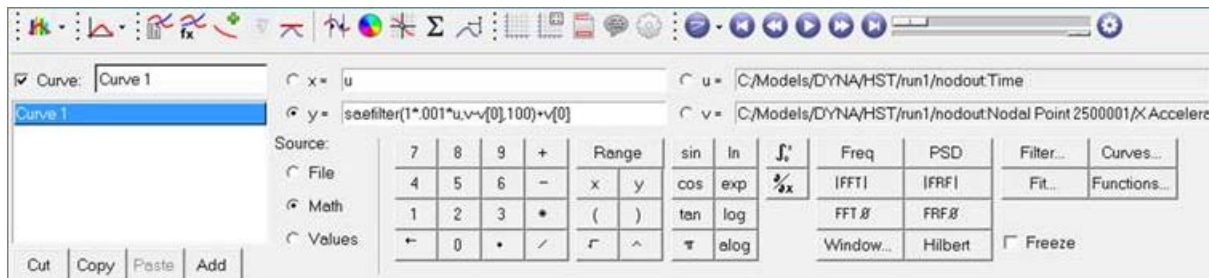
The **Advanced Plot Options** dialog contains a tree on the left side and the **Properties Table** on the right. From the tree, you can select the plot entity that you want to apply to the curve. The **Properties Table** lists the properties and current values for the selected plot entity.

Note: A curve must be specified in the **Build Plots** panel before you can apply options to it using the Advanced Plot Options feature.

Math on Curves Prior to Plotting

The HyperGraph **Math on Curves prior to plotting** feature allows you to use additional curve vectors to plot a curve. In addition to the traditional X and Y vectors, you will now be able to make use of U and V vectors to perform operations on a curve at plotting time. Thus, only one curve is generated in the session whereas in the older versions of HyperGraph, this could not be done without an initial curve.

To access the **Math on Curves prior to plotting** feature, select the **Define Curves** panel.



This is helpful any time you want to create a curve and immediately apply a function (for example, a filter) to that curve.

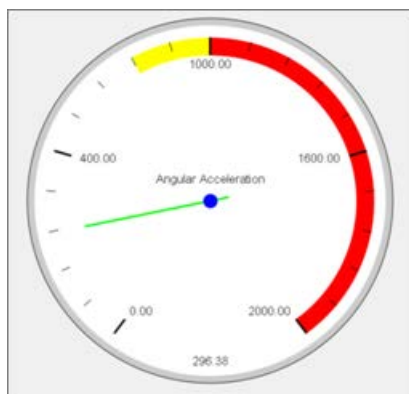
Previously, you had to create and plot the base curve, and then create subsequent curves which referenced the first (base) curve to perform the functions. The new **Math on Curves prior to plotting** approach simplifies this.

When you click **Apply**, the specified curve is created, using the vectors U and V, plus any math applied using X and Y.

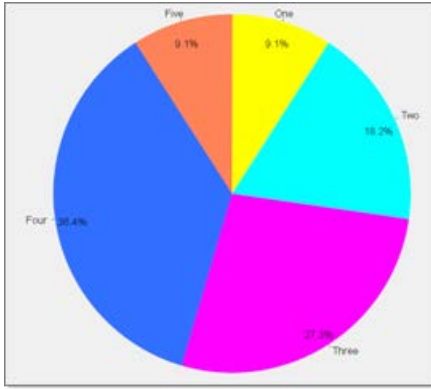
Custom Plot Types

Plots can be customized to be displayed as other plot types, such as Gauge, Pie, 3D Bar and more. Scripts have been included in HyperGraph 2D to display these custom plot types. Two of these scripts contain context sensitive menus to activate the override option for the plot window.

From an XY plot type, right-click in the plot window to display the context menu. From the **Custom Plot Types** context menu option, select **Gauge**. The Gauge plot type supports most XY features, such as Max/Min, Fixed or Scientific Notation, two Datum Lines (Red and Yellow only), fonts and more. Some XY plot features are not supported. This feature is designed for the first curve in a window only. See the example images below:



From a bar chart, you can right-click in the plot window to display the context sensitive menu. From the **Custom Plot Types** context menu option, select **Pie Chart**. The Pie Chart plot type supports most bar chart features such as Datum Lines, Fonts, syntax and more. Some bar chart features are not supported. This feature is designed for a single catalog in a window. See the example image below.



These scripts can be found in the [installation directory]/utility/scripts/plotting/overrides. These scripts can easily be modified to generate other custom plot types.

To turn off the custom plot type, right-click in the plot window, select the **Custom Plot Types** menu option and click **None**.

Resolved Issues

- Cross plotting animation files: Fixed the bug with cross plotting Data Types with animation files
- Axis Auto Fit has been corrected. To fix an axis, uncheck Auto Fit.
- Circle zoom has been fixed when working in the Coordinate Info panel
- Memory leak upon loading session file multiple times
- Proper reading of PLT files
- Linux: After using Export > Curves, user can't expand another menu without escaping from previous one first
- Improved unv reader
- 9.0 session file are fixed

Known Issues

- PLT files are not reading the data types properly, and you can get the wrong list in the component fields.
- Reader Versioning is not working correctly and DLLs require a core fix.

MotionView 13.0 Release Notes

MotionView is a user-friendly and intuitive multibody systems modeling environment. Its built-in parametric modeling capability and open architecture allows you to quickly build, analyze, and improve mechanical system designs even before physical prototypes are available. Coupled with a next generation solver, MotionSolve, MotionView provides the perfect solution for your multibody simulation needs.

This document describes changes that have occurred to MotionView since February 2013. All of these improvements are available in HyperWorks 13.0. The document is split into three parts:

- Core multibody functionality
- Automotive extensions to MotionSolve and MotionView
- The Altair Bushing Model

Core MotionView Functionality

3D Spline

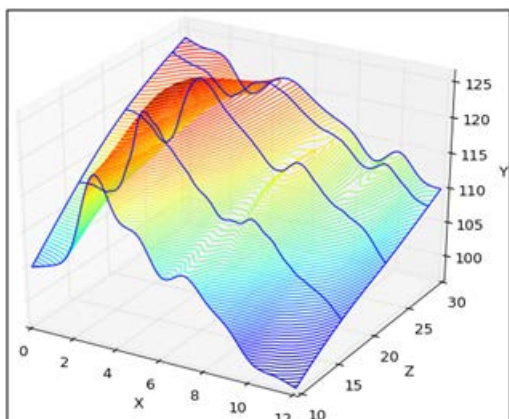
MotionView now supports 3D spline which can be used as reference data for forces and motions. A 3D spline contains several Y vectors at different Z planes, for a given X vector. It is an extended version of the 2D spline (defined as Curve – 2D Cartesian in MotionView). A typical application of 3D spline is in applying a combustion force in an internal combustion engine, where the force v/s crank angle data is available for different sets of engine speed. The force definition uses 2 independent variables (one for crank angle and other for engine speed).

A new icon is available in MotionView toolbar to add a 3D spline.

The 3D spline can be defined in two ways. An external .csv file with a specific format can be used as reference or the values in the form of the table can be manually entered in the panel.

Z	X	90.0000	120.0000	160.0000
		Y[0]	Y[1]	Y[2]
0	0.0000	100.1284	120.0585	95.2278
1	1.0000	100.1272	120.0580	95.2244
2	2.0000	100.1260	120.0575	95.2227
3	3.0000	100.1248	120.0575	95.2227
4	4.0000	100.1248	120.0570	95.2210

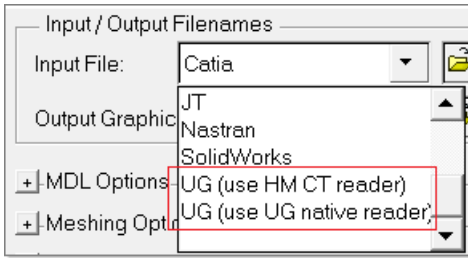
The **Show Spline** button in the panel brings up the visualization of the data.



Refer to > MotionView Panels> 3D Spline Panel for more details.

CAD Import Enhancements

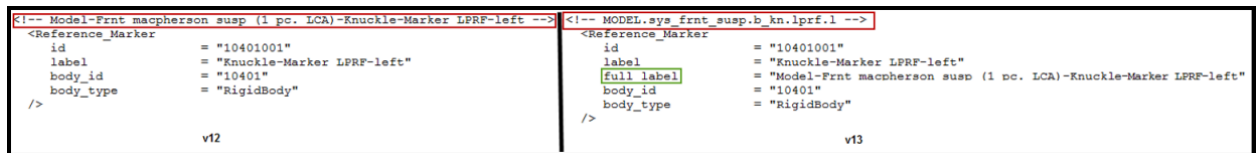
MotionView now supports using the new CT reader in HyperMesh. The CAD import utility now has 2 options for UG import. A faster import time is observed while using the new reader compared to the existing UG native reader.



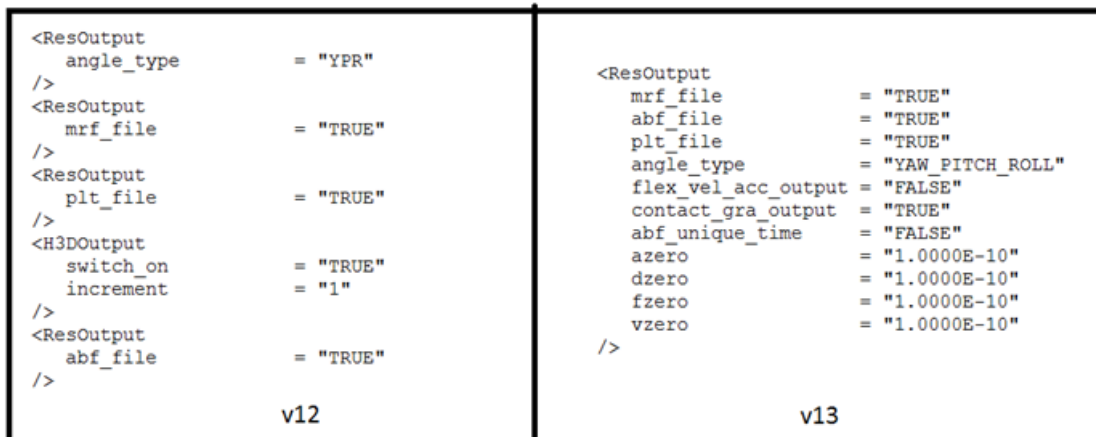
Changes in MotionSolve XML

There is a minor change in the way the entities from MotionView are exported into MotionSolve XML:

- The comment preceding the statement which contained the MotionView full label is replaced by MotionView varname
- The full label has been moved to an attribute within the statement block



- The several ResOutput statements in the Command section is now consolidated into one block



Flex Body Enhancements

Dummy bodies are now automatically added while exporting flexible body to MotionSolve when there is an offset between the interface node and corresponding marker. The dummy body is included when the offset exceeds the specified tolerance.

Product of inertia sign would appear reversed in Body Deformable panel compared to previous versions. This change makes display of inertia information in this panel consistent with the convention prevalent in MBD solutions.

Joint Friction for MotionSolve

MotionView now supports static and dynamic friction in joints defined for MotionSolve. Friction properties can be defined for Revolute, Translational, Spherical, Universal and Cylindrical joints. Symmetry is supported. MotionSolve uses the well-known LuGre friction model for these joints.

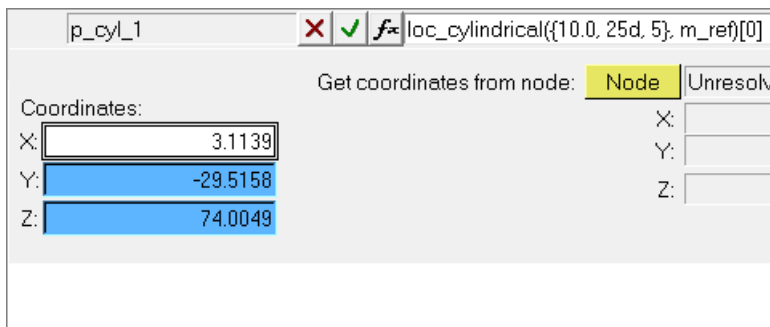
To learn more about joint friction support for MotionSolve, refer to the MotionView User Guide at MotionView > MotionView Panels > Joints Panel > Joints Panel – Friction Properties Tab.

Location Functions

MotionView 13.0 introduces new set of functions that can be used to define the Point entity. These functions help define points locations that are dependent on other coordinate systems and inputs, that generally could have resulted in complex math expressions. The following table lists the new functions with a brief description.

Function	Description
loc_rel_to	Locates a given set of coordinate values or point or marker in a given reference frame
loc_cylindrical	Locates a given set of coordinate values in a cylindrical reference frame
loc_along_dir	Locates a point along a vector at a certain distance from an origin point
loc_between	Locates a point between two points at a certain distance ratio
loc_mirror	Locates a point symmetric to a reference frame about a specified plane

These location functions return a vector of 3 values that represent the point coordinates in the global reference frame. Use the index [0], [1] or [2] to extract the x, y, z values respectively. An example of defining a point about a marker in cylindrical coordinates using the function “loc_cylindrical” is shown below.



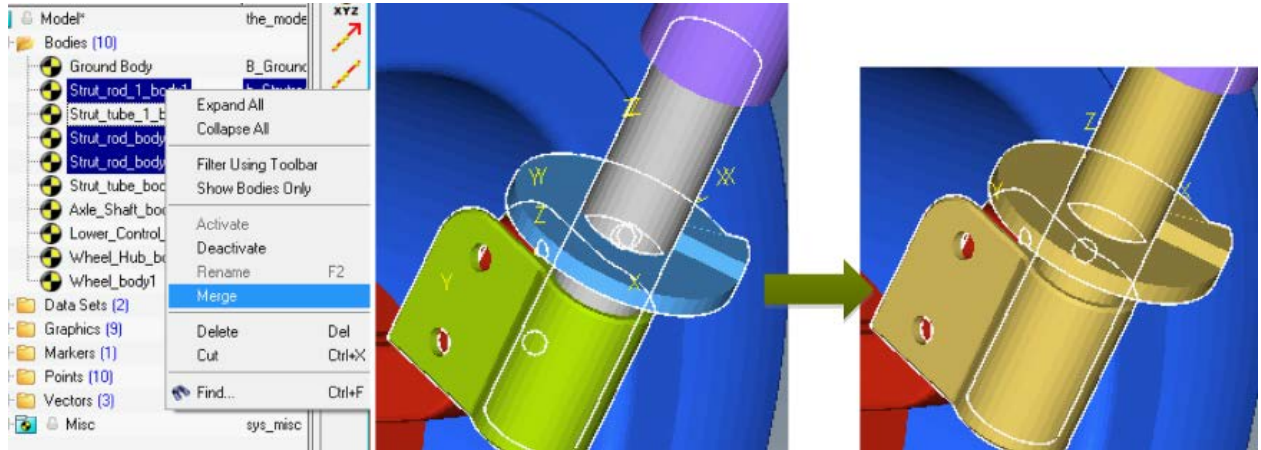
See the online help for more information.

Merge Bodies Feature

A new feature **Merge** is available now in the Project browser context menu that helps in merging two or more rigid bodies. This feature would be particularly useful during initial set up phase of model building, generally following a CAD import. Two or more bodies that may be represented as a single rigid body can be merged into one body using this feature. The merge handles:

- Calculation of combined mass/inertia and CG




- Changing other entity references (graphics, joints, and so on) from bodies being merged to the new bodies



- Merged bodies cannot be used on a pair body, point mass body or a deformable body
- Refer to *Merge Bodies* help to learn more about this feature

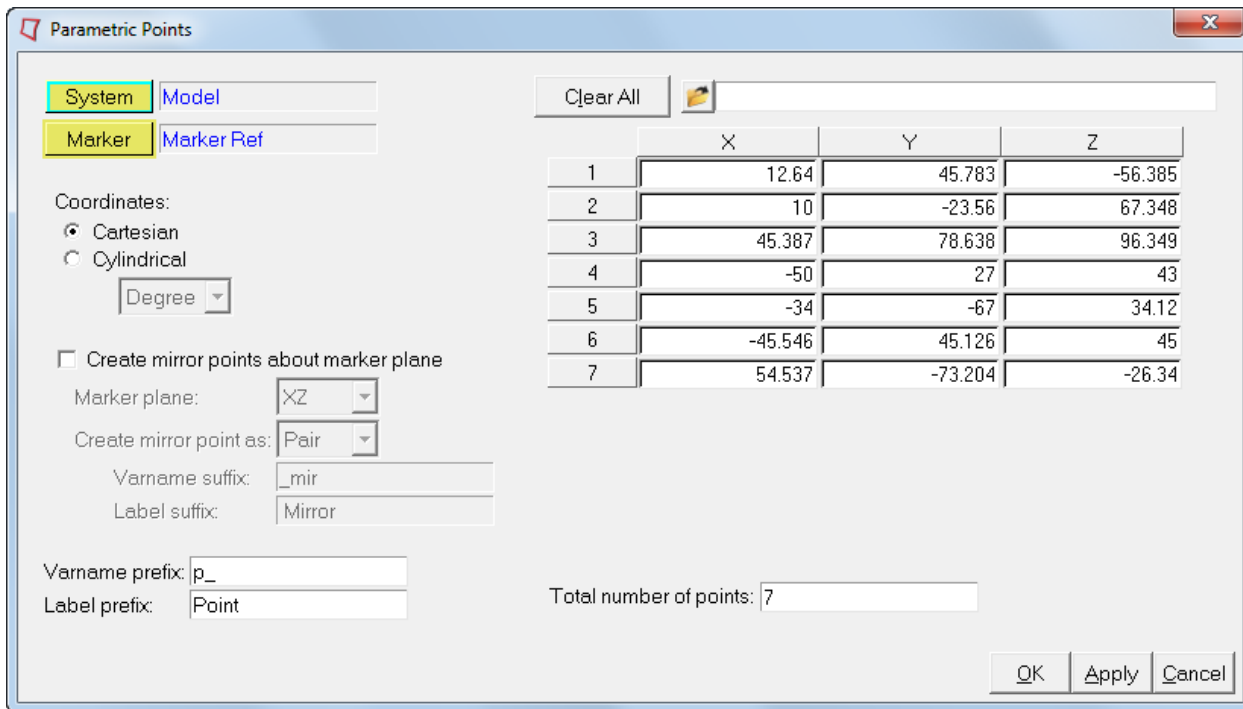
New Toolbars


To improve usability and ease of access new toolbars have been introduced.

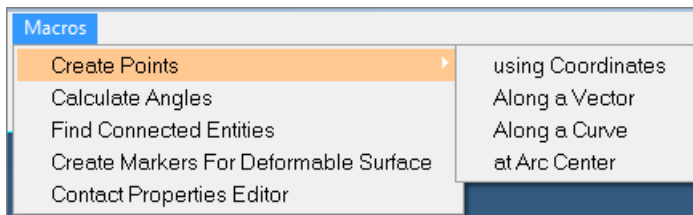
Point Macros		Helps to access Create Points macros – using Coordinates, along a Vector, along a curve and at Arc center
Other Macros		Provides access to other macros – measure angles, Find Entities, Create Markers on Deformable surface and Contact Properties Editor
Check Model		Available on the MotionView toolbar, helps in executing “Check Model”

Point Creation Utility

A new point creation utility called **Parametric Points** is now available. This utility helps in importing large set of points, with reference to different set of coordinate systems. The utility can be used to create points in cylindrical coordinates as well as create mirroring points with respect to the defining coordinate systems. The Parametric Points utility utilizes the location functions described above.



The utility can be accessed from the Utility menu by clicking **Create Points > using Coordinates** or through the  icon in the Point Macros toolbar (newly introduced in this release) .



Refer to MotionView Menus > Macros Menu > Create Points > using Coordinates in the MotionView help for more details.

Project Browser Enhancements

Performance improvements:

- Faster response times when opening models and updating the browser. For example, a full-vehicle model with sixteen analyses now opens 15-35% faster than prior releases.
- Performance bottlenecks in the Browser's 'Delete' operation have also been removed. The time it takes for the 'Confirm Delete' pop-up to appear is much quicker now, especially when the number of items being deleted is large.
- Color changes made through the browser now propagate more quickly than in previous releases; we have seen 30-50% faster times when cascading a color from parent to its children using this release.

Progress reporting: The status area below the Project Browser now reports progress when opening models with messages:

- 'Loading Model...'
- 'Populating the browser...'

New keyboard shortcuts: Additional keyboard shortcut for the Project Browser have been enabled. (Refer to the Online Help for other supported keyboard shortcuts.)

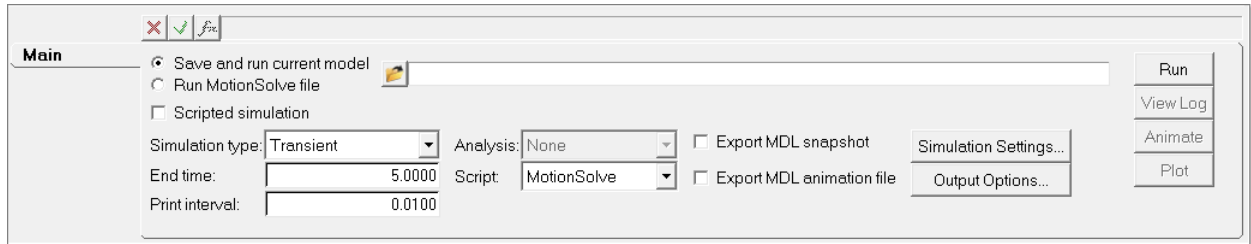
- Ctrl+A is enabled to 'Select All' entities in the Project Browser.
- Ctrl+\ is the shortcut for 'Unselect All' entities

- Shift+Delete now allows you to delete immediately, without being prompted by the 'Confirm Delete' dialog.

Run Solver Panel Enhancements

The Run Solver panel for MotionSolve solvermode has been revamped to simplify and improve usability.

- The various dynamic tab has been removed
- The Simulation End time and Print interval is moved to the Main tab along with Simulation type
- Simulation Parameters are available in Simulation Settings dialog
- Output Options are available in Output Options dialog
- Check Model button has been removed. Instead a Check Model icon is available in the MotionView toolbar which can be used at any instance (not just when Run panel is active)



- Simulation Settings and Output options dialog supports wider range of attributes for MotionSolve
- View Log button is available to view the log file of the solver run

Shared Definitions

Entities such as Systems, Assembly and Analysis, (referred to as definition based entities) are created through an instantiation statement in the MDL that refers to a definition block. It is possible that multiple instances can be instantiated off of the same definition block. Such instances are referred to as shared instances or linked instances. Below is an example model containing shared instances.

```
*BeginMDL( the_model, "Model", )
*System( sys_0, "System 0", def_sys_0 )
*DefineSystem( def_sys_0 )
  *System( sys_1_shared, "System 1", def_sys_1, p_0, p_1 ) // sys_1_shared uses definition def_sys_1
  *System( sys_2_shared, "System 2", def_sys_1, p_2, p_3 ) // sys_2_shared also uses the same definition def_sys_1
  *Point( p_0, "Point 0" )
  *PointPair( p_1, "Point 1" )
  *Point( p_2, "Point 2" )
  *PointPair( p_3, "Point 3" )
  *SetPoint( p_0, 0.0, 1.0, 10.0 )
  *SetPoint( p_1, LEFT, 0.0, -10.0, 0.0 )
  *SetPoint( p_2, 0+10, 1.0, 10.0 )
  *SetPoint( p_3, LEFT, 0.0+10, -10.0, 0.0 )
*EndDefine()
*DefineSystem( def_sys_1, p_att1, p_att2 ) // Definition block def_sys_1 begins
  *Attachment( p_att1, "Attachment point 1", Point, "Select attachment.", , )
  *Attachment( p_att2, "Attachment Point 2", PointPair, "Select attachment.", , )
  *Point( p_in_shared, "Point in shared def" )
  *Body( b_0, "Body 0", p_att1, , , )
  *Graphic( gsph_0, "Sphere 0", SPHERE, b_0, p_att1, 2.0 )
  *BodyPair( b_1, "Body 1", p_att2, , , )
  *GraphicPair( gbox_0, "Box 0", BOX, b_1, CENTER, p_att2, ZX, VECTOR, V_Global_Z, VECTOR, MODEL.V_Global_X, 2.0, 2.0, 2.0 )
*EndDefine() // Definition block ends
*EndMDL()
```

```
*BeginMDL( the_model, "Model", )
  *System( sys_0, "System 0", def_sys_0 )
  *DefineSystem( def_sys_0 )
  // sys_1_shared uses definition def_sys_1
  *System( sys_1_shared, "System 1", def_sys_1, p_0, p_1 )
```

```

// sys_2_shared also uses the same definition def_sys_1
  *System( sys_2_shared, "System 2", def_sys_1, p_2, p_3 )    *Point( p_0, "Point 0"
)

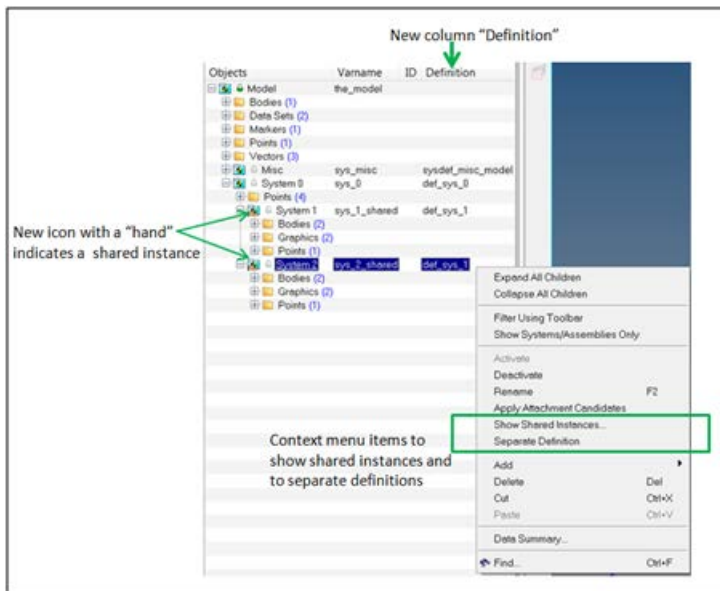
  *PointPair( p_1, "Point 1" )
  *Point( p_2, "Point 2" )
  *PointPair( p_3, "Point 3" )
  *SetPoint( p_0,                                0.0, 1.0, 10.0 )
  *SetPoint( p_1,                                LEFT, 0.0, -10.0, 0.0 )
  *SetPoint( p_2,                                0+10, 1.0, 10.0 )
  *SetPoint( p_3,                                LEFT, 0.0+10, -10.0, 0.0 )
*EndDefine()

// Definition block def_sys_1 begins
*DefineSystem( def_sys_1, p_att1, p_att2 )
  *Attachment( p_att1, "Attachment point 1", Point, "Select attachment.", , )
  *Attachment( p_att2, "Attachment Point 2", PointPair, "Select attachment.", , )
  *Point(p_in_shared, "Point in shared def")
  *Body( b_0, "Body 0", p_att1, , , , )
  *Graphic( gsph_0, "Sphere 0", SPHERE, b_0, p_att1, 2.0 )
  *BodyPair( b_1, "Body 1", p_att2, , , , )
  *GraphicPair( gbox_0, "Box 0", BOX, b_1, CENTER, p_att2, ZX, VECTOR, V_Global_Z,
VECTOR, MODEL.V_Global_X, 2.0, 2.0, 2.0 )
  *EndDefine()
// End Definition block
*EndMDL()

```

MotionView 10.0 GUI and earlier versions handled shared instances as though they originated from different definitions. Once the model with shared instances loaded in MotionView and any of the shared instances got modified, the linkage was severed and MotionView saved the instances by referring them to different definitions. This inherent behavior was lost from version 11.0 onwards. For examples, modifying a system instance by adding, deleting or changing the topology of an entity within, resulted in the same change being applicable to the other instance that shared the same definition as the system

From version 12.0.110, better support for shared instances has been provided in the MotionView GUI.



- The browser indicates the sharing between instances with a separate hand icon appended to the standard icon of the entity.
- The user has a choice of breaking linkage automatically when modifying a shared instance. A new option is available under Tools -> Options -> Build Model to control how shared instances should be handled while modifying them
- Right click context menu in browser provides option to list shared instances as well as stop sharing the definition

To learn more about managing shared definitions, refer to the MotionView User Guide at MotionView > MotionView Overview > Container Entities > Managing Shared Definitions.

Other Enhancements

- The Mesh and Shade options in the Visualization tab of Graphics entities are reorganized to streamline with the possible options available:



- Model Selection Behavior: Selecting the model from the Project (or using **Model > Select** from the graphics window) no longer highlights everything in the OpenGL window. This helps to avoid the 'cluttered' look of having everything in the model shown selected in white. Note that the panel area still updates to show the System panel corresponding to the model, as expected.
- MotionView templex functions now supports appending 'd' as a keyword to suggest angles being in degrees

Resolved Issues

Entities

- Mass, inertia, CM were not being written to adm when CM Coordinates was not assigned for the body. This issue is fixed.
- Issue of missing orientation information of a marker belonging to a Point mass body is fixed.
- Direction Cosine of markers were displayed incorrectly when using "Orient by Angles" option. This issue is resolved.

- There was an issue with the point collectors for orienting markers not selecting an attachment point of the parent system. This issue is now resolved.
- An application error related to toggling of a marker as Floating marker is resolved
- A bug in reporting missing reference files is fixed. The "File not found" error was not being reported the second time or if there were more references missing in the Curve panel
- An application error on Export Curve in the Curve panel is fixed
- Origin field in the Joints panel not appearing with blue background while referring to the Point pair attachment is fixed
- Explicit Spring graphics can now be added while defining a spring damper
- Curve Panel: Clicking on "Show curve" with Interpolation fails with errors when HW session contains a HyperGraph client. This issue is fixed
- "Show curve" failed to update once an erroneous value has been entered in the curves panel. This issue is fixed
- Flickering of HyperWorks application was observed when reading curve data with display set with small fonts. This issue is resolved
- Access to help on Vector panel through F1 key was broken in v12.0. This is fixed
- An issue related to erroneous setting of isbush datamember for non-compliant joints during cut & paste into a system or analysis is fixed
- Curve to Curve joint definition between a flexible body and rigid body resulted in error in MotionSolve. This issue has been resolved
- An issue of shell (.shl) graphics not being read by MotionView is fixed
- "Material inside" information related to box, cylinder and sphere graphics is not saved into the mdl file. This issue is fixed
- An application error on using "Orient by angles" option to orient Beam I or J markers is fixed
- The Rho (r) property in polybeam now displays value in exponential form
- Parametric expressions in the Solver String entity field were not being evaluated. This issue is resolved
- Solver Differential equation that contained Static Hold and Implicit option resulted in an additional comma (,) between these arguments during an ADM export. This issue has been resolved
- An issue of missing <Post_request for compliant joint in MotionSolve xml when the output is created by Output on Body in MotionView is resolved
- An issue of MotionSolve XML containing <Post Request related to compliant joint pairs when the joints are set to non compliant is resolved
- An error encountered when cutting a Template and pasting into another System or Analysis, is fixed
- A resulting application error while modifying a Form entity within an Assembly is fixed
- A problem with ineffective deactivation of the Forms entity is fixed
- An application error is selecting dataset options after repeated clearing of MotionView session is fixed
- Add Assembly dialog ignored the Attribute "Type" while adding or reselecting assembly using existing data. This issue is resolved
- A problem with references to the left and right side of pair entities in the Attachment Candidates dialog is fixed
- The issue of incomprehensible Messages in message log related to deactivated entities is now fixed
- 'Cone' option for 'Tangent-Plane' type in Constraint Mate entity in Advanced Joints was missing. This issue is resolved

- Output panel for Action only Force on Body did not gray marker choice and set to I_MARKER. This issue is resolved
- Entity Info dialog, using note-pin was not refreshing ID field(s) when using 'Evaluate ID on Apply' option. This is fixed
- Plant Input and Plant Output references were not being written into <Param_Linear statement, with "Use in Linearisation" flag is checked on. This issue has been resolved
- Nonlinear bushings Displacement outputs were exported to MotionSolve referring to J Float marker. In certain scenarios, it resulted in erroneous measurements. This has been corrected. The output is now measured between the I and RM markers
- The attribute "hold_order" for Control_PlantInput was not being exported to MotionSolve XML. This issue has been resolved.
- U and V parameters of parametric surface were not exported to MotionSolve. This issue is fixed
- Surface graphics entities were not being exported to MotionSolve. This issue is fixed
- An error in creating deformable surface due to misinterpretation of node values specified in exponential form is resolved

CAD Import

- Importing Parasolid resulted in incorrect scaling of graphics. This issue has been resolved.
- Mass, Inertia scaling was not proper when a FE model in unit SLINCH were imported. This issue has been fixed.
- Issue of Parasolid imports that results in several parts being combined into fewer bodies is resolved.
- Issue related to failure of import while specifying Locator points has been resolved.
- Parasolid Import will henceforth automatically set "Input file length" as meters.
- Points were not being created during CAD/FE model import with interactive HyperMesh option. This issue is fixed.
- An application error while closing "Import is a success" message after clearing the message log is fixed.

Flex Bodies

- Copying a long list of node numbers in Interface nodes fields in Flexprep utility resulted in a scrambled view. This issue has been fixed.
- Issues related to incorrect modes and interface nodes while converting h3d to mnf and mnf to h3d are resolved.
- Erroneous GENSTIFF matrix was being written out while converting h3d to mtz when using kgf as the solver unit. This issue is fixed.
- MotionView now exports ModeLoad section to MotionSolve XML whenever the flex body h3d contains modal loads.
- Flexbody generation tool used to lock the .out file resulting in non-usability of the tool using the same output filenames. This issue has been resolved.
- Damping values in the Flexbody panel in Abaqus solver mode were not being populated on loading of the model. This issue is fixed.
- SUP file path was not getting populated in Body panel for Abaqus solver mode. This issue is resolved.
- Message when a dummy body was introduced upon export at flex body interface node did not include the corresponding marker's parent entity. This issue is fixed.
- Message related to insertion of dummy bodies for a flexible body is repeated while exporting to Adams.

- The Find Nodes dialog in Body Deformable -> Properties tab were not listing markers from compliant joint. This issue is now resolved.

Graphical User Interface

- Selecting a Point Mass body does not highlight its implicit graphic on graphic screen. This issue is fixed.
- An issue with scroll wheel not working in the Expression builder is resolved.
- "Run" toolbar was not visible when a folder with name "run" existed in the model directory. This issue is resolved.
- Entity attachment collectors for a system failed to pick entities from the graphic area when a default was specified for the attachment. This issue is resolved.
- Expression builder reports "Error Evaluating" while accessing data members within a table. This issue is fixed.
- Expression builder does not show datasets under sys_misc system. This issue has been resolved.
- Entity selected in browser of a MotionView window is lost when changing between multiple MotionView windows. This issue is fixed.
- Right click at the bottom of the browser resulted in context menu to appear further away in Linux. This issue has been resolved.
- Selection focus in the Project Browser was lost when an entity was deleted. This issue is resolved.
- Project Browser: Find tool or search feature returns blank entries such as in Datasets ID column when searching for '0'. This issue is resolved.

MDL

- Graphic system definition failed to import using the System -> Import tab. This issue is resolved.
- An app error in the task wizard when more than one task has the same varname is fixed.
- Paths for definition files of auto-entities were being saved as absolute paths. This issue is fixed.
- MotionView crashes when opening a model which does not specify "varname" in *BeginMDL() block. This issue is fixed.
- Orientation references of marker in an assembly were saved as global reference (e.g. MODEL.body.cm.xaxis) instead of as a local reference (body.cm.xaxis). This issue is fixed.
- Warning message appearing due to illegal pair designation for graphic definition is resolved. The warning used to appear when MotionView corrects the graphic instantiation using *GraphicPair statement instead of *Graphic statement.
- CommandSet now supports deactivation and activation of VFORCE/VTORQ and GFORCE elements. The deactivation of pair entities, while exporting to ACF file, was ignored. This issue is fixed.

Open/Save/Import/Export

- Importing of adm statements with user subroutine having a space between USER and following bracket resulted in an incomplete expression. This issue is fixed.
- ADM Import: Function with value in a MOTION statement resulted in an error. This issue is fixed.
- ADM files with expressions having large blank spaces from the beginning of the line leads to expressions being truncated.
- A system with an existing definition name being imported in an assembly renames the definition. This issue is fixed.
- Problem with *Set statements not being saved to the data file mdl after cut and paste into an assembly is resolved.
- *RegisterSolverScript and *RegisterReportsLog specified in a user preference is not recognized by MotionView. This issue is resolved.

- MotionSolve XML export contained compliant joint markers when joints were set to non-compliant. This issue is resolved.
- An application error encountered when exporting data in "Data Summary" dialog, is fixed.
- There was an issue with the Save As dialog on Assembly not appending .mdl extension automatically. This issue is fixed.
- Loading a model with assemblies having two instances of an external definition resulted in a crash. This issue has been resolved.
- Saving of mdl file in Linux platform sometimes resulted in creation of a null file. This issue has been resolved.
- Browsing for file name from File fields on Linux and Mac shows full path in the file select dialog instead of file name only.
- The issue of mdl_batch not accepting space in the path of input file has been resolved.
- The body and graphic entity information within an assembly was not being written to MAF file. This issue has been resolved.

Performance

- Opening Assembly models with several sub assemblies took longer times compared to cumulative time for opening individual sub assembly models. This issue is resolved.
- Issue of longer times on opening models containing assemblies with shared definition is resolved.

Others

- The macros to create markers for deformable surface create incorrect location of markers when node coordinates were specified in exponential form. This issue is fixed.
- An Application Error while using right click in "Find Connected Entities" macro is fixed.
- Add' dialog's keyboard shortcuts don't work when the Caps-Lock is on. This issue is fixed.
- MV entity referring to a file in library via URL hwedm:// is shown as Error Evaluating in 12.0.110. This issue is fixed.
- Some of the Output options in the Run panel such as Multibody Results File (.mrf) and Altair Binary File (.abf) were not saved in MDL. This issue is fixed.
- The FIM_S selection of the Force Imbalance Method in the Static simulation parameters tab was exported as FIM_D. This issue is fixed.
- Data members such as state and ic_use_trans (Joints entity) in MotionView model were not accessible in HyperStudy. This issue has been resolved.
- Mass /moment of Inertia allowed adding the same component so many times. This issue has been resolved.
- Mass moment of inertia for Polybeam bodies was not listed in CG inertia summary. This issue has been resolved.
- CG inertia summary allowed adding the same body in the table more than once. This issue is fixed.
- Application error was observed in the CG Inertia Summary due to unresolved references in the Inertia Coordinates tab in the Body panel. This issue is fixed.
- A Java exception error in Load Export while extracting the "Peak" values of the load is resolved.
- Load Export now supports export of loads due to field entity.

Automotive Extensions in MotionView and MotionSolve

Altair MotionView ships with a vehicle library that can be assembled using the Assembly Wizard and Task Wizard dialogs. The library is made up of system definitions that can be assembled together to represent either a front half vehicle, rear half vehicle, full vehicle or power train. The Assembly Wizard allows you to select and assemble different permutations of a vehicle from the available wizard options. After creating an assembly wizard model, you can add an analysis using the Task Wizard dialog.

Vehicle Dynamics Tools

MotionView and MotionSolve support an extensive set vehicle simulation tools.



- MotionView now includes new vehicle dynamics preference. New entities are available for modeling bushings, springs, dampers, rebound-stops, bump-stops, tires and drivers
- MotionSolve contains new dlls (or shared libraries) that capture the behavior of these entities
- The tools support activities in the areas of:
 - Suspension kinematic and compliance (K&C) analysis,
 - Vehicle dynamics,
 - Durability, and,
 - Ride Comfort






These entities support TiemOrbit property files along with reuse of common Adams/Car™ property files with your MotionView and MotionSolve models.


The MotionView online help describes how to add these entities to your model. Use of the vehicle dynamics tools requires installation of HW-Solvers 13.0. 0.

In MotionSolve, the following are common to all modeling elements:

- An equal but opposite force is applied on two bodies that these entities are defined between
- The force applied depends on the displacement and velocity of one body relative to the other
- Outputs specific to each entity are supported, for example tire lateral slip angle for the tire entity
- The parameters (for example stiffness), used to compute the force are stored in a property file that is independent of the MotionView MDL file and the MotionSolve input deck. The property files are
 - Text files and may be viewed and edited using any common text editor
 - Compatible with Adams/Car – property files from Adams/Car may be reused with the Auto Entities in MotionView
 - Read by MotionSolve just prior to analysis

Entity	Function	Features
Bushing (autoBushing) 	Elastically mount suspension arms, links, sub-frames, engines, gearboxes, etc. to vehicle chassis	<ul style="list-style-type: none"> • Exerts force and torque in all directions. • Non-linear force/torque verses deflection interpolated from tabular data using Akima's method • Linear damping
Bump-stop (autoBumpstop) 	Limit suspension bump or jounce travel	<ul style="list-style-type: none"> • Exerts a force along a line to repel two bodies when the distance between the bodies is less than the threshold • Non-linear force verses deflection interpolated from tabular data using

		<p>Akima's method</p> <ul style="list-style-type: none"> • Linear Damping
<p>Damper (autoDamper)</p> 	<p>Dissipate energy stored in springs and damp chassis motions</p>	<ul style="list-style-type: none"> • Exerts a force along a line between two bodies • Non-linear force verses velocity interpolated from tabular data stored in a property file using Akima's method
<p>Rebound-stop (autoReboundstop)</p> 	<p>Limit suspension rebound travel</p>	<ul style="list-style-type: none"> • Exerts a force along a line to attract two bodies when the distance between the bodies exceeds a threshold • Non-linear force verses deflection interpolated from tabular data using Akima's method • Linear Damping
<p>Spring (autoSpring)</p> 	<p>Limit forces in suspension components and isolate chassis by absorbing energy from vertical road inputs</p>	<ul style="list-style-type: none"> • Exerts a force along a line between two bodies. • Non-linear force verses deflection interpolated from tabular data using Akima's methods • Optionally set the initial load or length of the spring at vehicle input position
<p>Tire - FTire Version 14-2 (autoTireCTI)</p> 	<p>Generate forces for steering, accelerating, and braking. Isolate the vehicle from small road irregularities</p>	<ul style="list-style-type: none"> • COSIN FTIRE (flexible ring) • FIALA • Pacejka 2000 Magic Formula tire models • 3D and 2D Road Models including Curved Gridded Road (CRG) • COSIN RGR • The version 14-2 of FTire provides improved handling of 3D roads • An improved Bekker soil model is now supported • There are many other enhancements. For more information regarding FTire, please see http://www.cosin.eu/release_notes
<p>Tire – CDtire Versions 3.0.3 and 4.0 (autoTireCD)</p> 	<ul style="list-style-type: none"> • Generate forces for steering, accelerating, and braking • Isolate the vehicle from small road irregularities 	<ul style="list-style-type: none"> • Supports family of CD-Tire models (flexible ring) • 2D and 3D Road Models including CRG • An environment variable CDTIRE_VERSION specifies the version to be used. If the environment variable is not set then CDtire Version 4.0 is used. If the environment variable is set to "303", then CDtire Version 3.0.3 is used. • CDtire version 4.0 supports two new

		<p>tire models CDT30/RT and CDT50</p> <ul style="list-style-type: none"> • CDT30 is a real-time compatible tire model. In CDT30, arbitrarily small road wavelengths may be specified in tire plane, but the profile is considered constant in the lateral direction • CDT50 is capable of supporting full 3D roads and 3D obstacle enveloping. CDT50 provides highly accurate answers • These 2 models are only supported in transient simulation • User Defined Road & RSM4000 (3D road) are now fully supported
<p>Tire – TNO Version 6.2 (autoTireTNO)</p> 	<ul style="list-style-type: none"> • Generate forces for steering, accelerating, and braking. • Isolate the vehicle from small road irregularities 	<ul style="list-style-type: none"> • MF-TYRE – Pacejka 2008 • SWIFT-TYRE (rigid-ring) • 2D and 3D Road Models, including CRG • MF-SWIFT Version 6.2 includes a motorcycle tire model that includes high inclination angles characteristic of motorcycle tires • Version 6.2 supports OpenCRG release 1.0.5 for including measured 3D roads in a MotionSolve simulation • NOTE: The licensing of MF-SWIFT is different in version 6.2. MF-SWIFT users must obtain a new license file and new license server software from TNO. MF-TYRE is not affected by this change

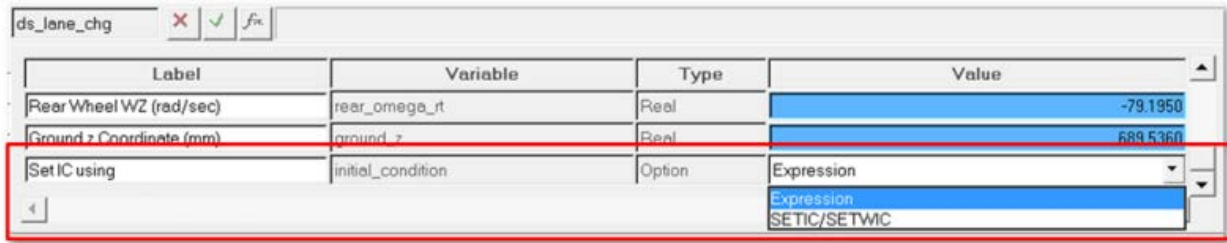
To learn more about auto entities, refer to the MotionView User Guide at MotionView > Vehicle Dynamics Modeling > Auto Entities.

Vehicle Initial Conditions

This new feature enables the solver to assign initial velocity conditions to a full vehicle model during run time. This also enables the solver to predict the initial conditions of the rotating wheels based on the loaded radius that is obtained during the static analysis. This makes the initial conditions provided to the transient simulation more accurate compared to the old method of setting the initial conditions explicitly using the unloaded radius.

MotionView – MDLLIB option

The library now provides an option in the analysis dataset to use a user control subroutine CONSUB to set the initial conditions of the vehicle.



MotionSolve - CONSUB

SETIC sets the vehicle longitudinal velocity and SETWIC sets the rotational velocities at the wheels.

SETIC

Function name: SETIC

DLL name: mbdtire

User Expression: USER(<Branch ID>, <Reference marker ID> , <Vx>, <Vy> , <Vz>)

Feature notes: Assigns appropriate initial velocity to the bodies

Parameter	Description	Type and Range	Dimension
Branch ID	ID to indicate the origin of the mode	{ Integer } 635 : MOTIONSOLVE	NA
Reference marker ID	Reference marker ID	{ Whole number }	NA
Vx	Velocity along x axis of the reference marker in mm/s	{ Real number }	LT-1
Vy	Velocity along y axis of the reference marker in mm/s	{ Real number }	LT-1
Vz	Vz	{ Real number }	LT-1

Example in MotionSolve xml Deck:

```
<UserProgramControl
  usrsb_param_string = "user(100,30101010,19999.7)"
  usrsb_dll_name     = "mbdtire"
  usrsb_fnc_name     = "SETIC"
/>
```

SETWIC

Function name: SETWIC

DLL name: mbdtire

User Expression: USER(<Branch ID> , <VM>)

Feature notes: Assigns appropriate angular velocity to wheels

Uses loaded radius of the tires

Parameter	Description	Type and Range	Dimension
Branch ID	ID to indicate the origin of the mode	{ Integer } 635 : MOTIONSOLVE	NA

VM	Magnitude of Initial velocity in SETIC in mm/s $\text{SQRT}(V_x^2 + V_y^2 + V_z^2)$	{Positive real number}	LT-1
----	--	------------------------	------

Example in MotionSolve xml Deck:

```
<UserProgramControl
  usrsub_param_string = "user(635,19999.7)"
  usrsub_dll_name     = "mbdtire"
  usrsub_fnc_name     = "SETWIC"
/>
```

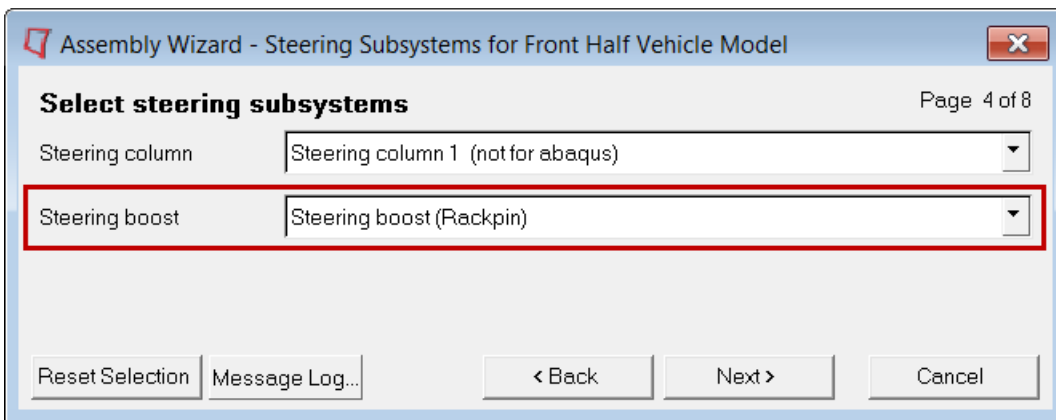
Hydraulic Power Steering Model (Rack and Pinion)

The power steering for the rack and pinion is completely redone and simulates the parts of a hydraulic system with inputs for boost cylinder area and boost curve.

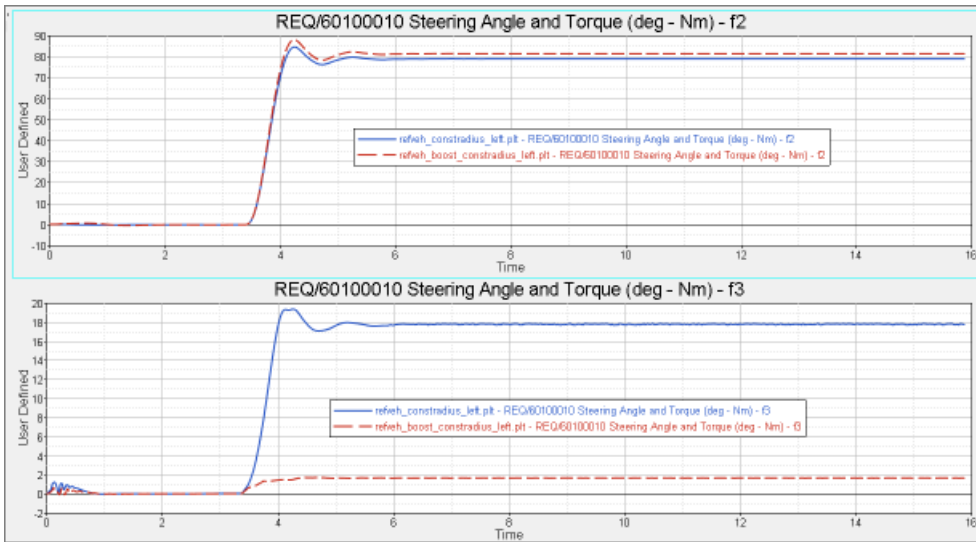
Features:

- Uses a circular beam instead of a linear spring to connect the gear input shaft to the pinion
- User inputs are boost cylinder dimensions and boost curve in PSI
- Boost force is applied at the rack and is a function of
 - Area of cylinder,
 - Angular twist (valve opening) in the beam,
 - Boost curve,
 - Cylinder efficiency in %

This new system can be activated from the MDLLIB using a new option that is available.

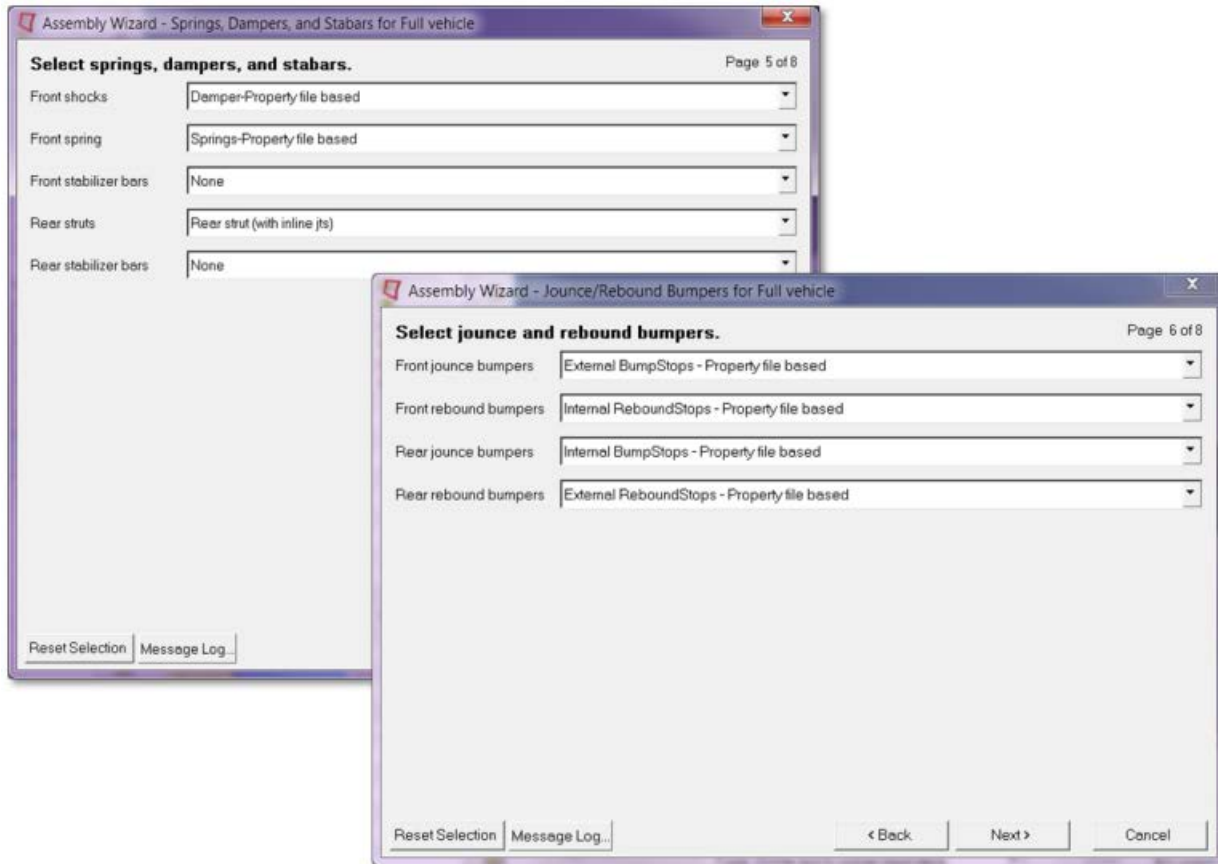


The steering boost would reduce the amount of steering effort required to turn the steering wheel.



Adding Property File Based Systems (AutoEntities)

The Assembly Wizard in 13.0 now allows you to add a system definition that uses a property file for the springs, shocks, bumpstops and reboundstops. The system definitions display their own panel when selected if the Vehicle Dynamics Tools preference is loaded.



Frequency Dependent Bushing Model

MotionAuto provides a new extension to MotionSolve for modeling the amplitude and frequency dependent behavior of rubber and fluid filled (hydro-mount) bushings. The amplitude and frequency dependent behavior is important for ride-comfort and durability simulations. The new model's features include:

- Frequency and amplitude dependent rubber and hydromount damping formulations

- Constant, cubic, and spline stiffness formulations
- Local structural stiffness and travel limits of the bodies the bushing connects
- Input (displacement and velocity) scaling
- Output (force and torque) scaling
- Force and torque preloads
- Displacement offsets
- Friction and stiction based on the LeGrue model

HyperGraph includes a Model Identification Tool (MIT) which given test data identifies bushing model parameters, plots model results against test results, and writes a TeimOrbit™ format .gbs file containing the model parameters. MotionSolve reads the bushing parameters from the .gbs file prior to analysis.

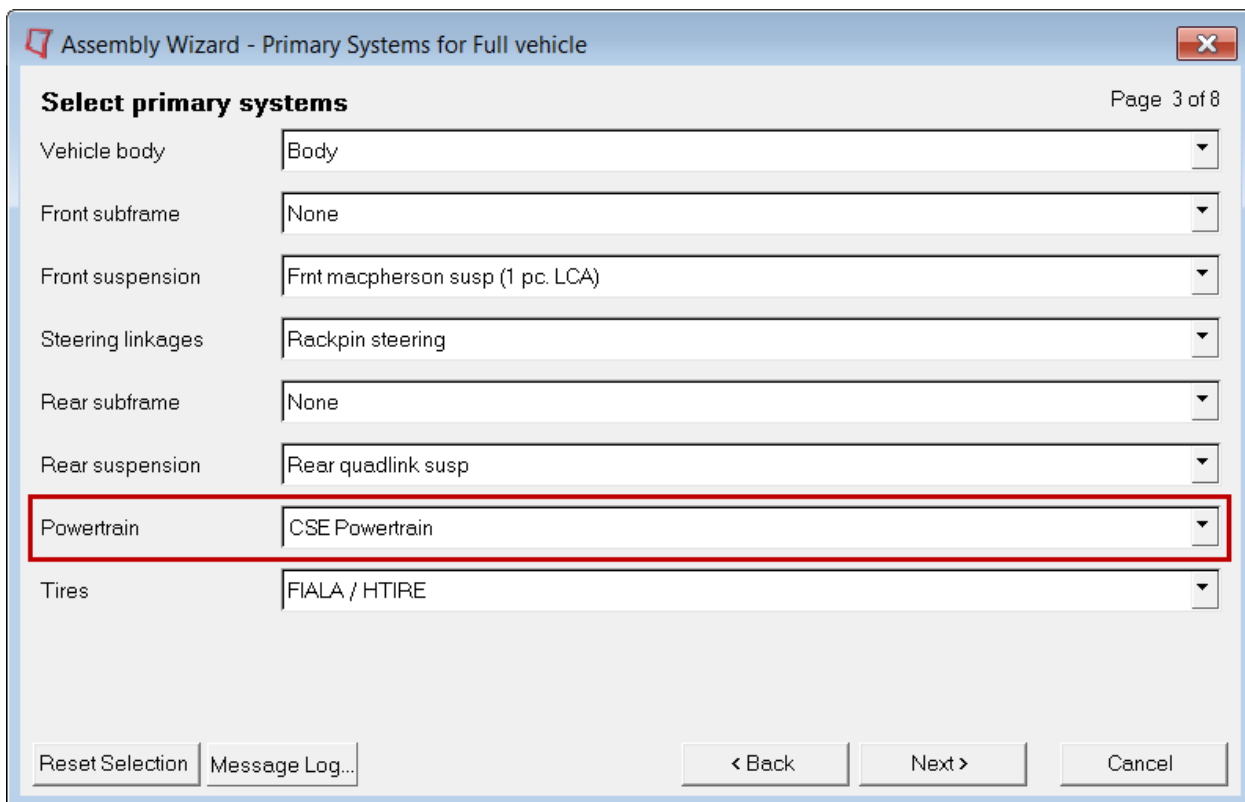
MotionView 's MBD -Vehicle Dynamics Tools includes support for adding frequency dependent bushings to a MotionView model; second, For more information see:

- HyperGraph MIT documentation
- Vehicle Dynamics Tools documentation for frequency dependent bushing
- Simulation Model documentation

Advanced Drivers (Experimental)

CSE Powertrain Model

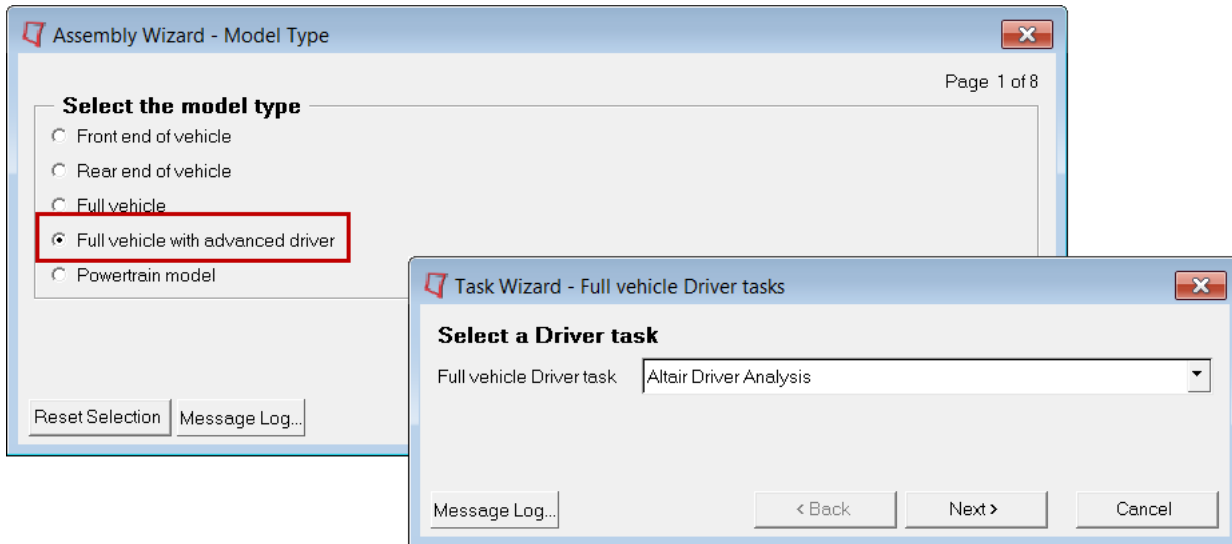
The 13.0 vehicle library includes a new control state based powertrain that consists of an IC engine, clutch and manual gearbox. It is available when assembling a full vehicle with the advanced driver.



Altair Open Loop Driver

MotionView 13.0 includes Altair's own driver analysis which currently supports open loop events only. Setting the environment variable `HW_MV_EXPERIMENTAL = ADVANCED_DRIVER`, enables an option in the Assembly Wizard to build a full vehicle with this advanced driver.

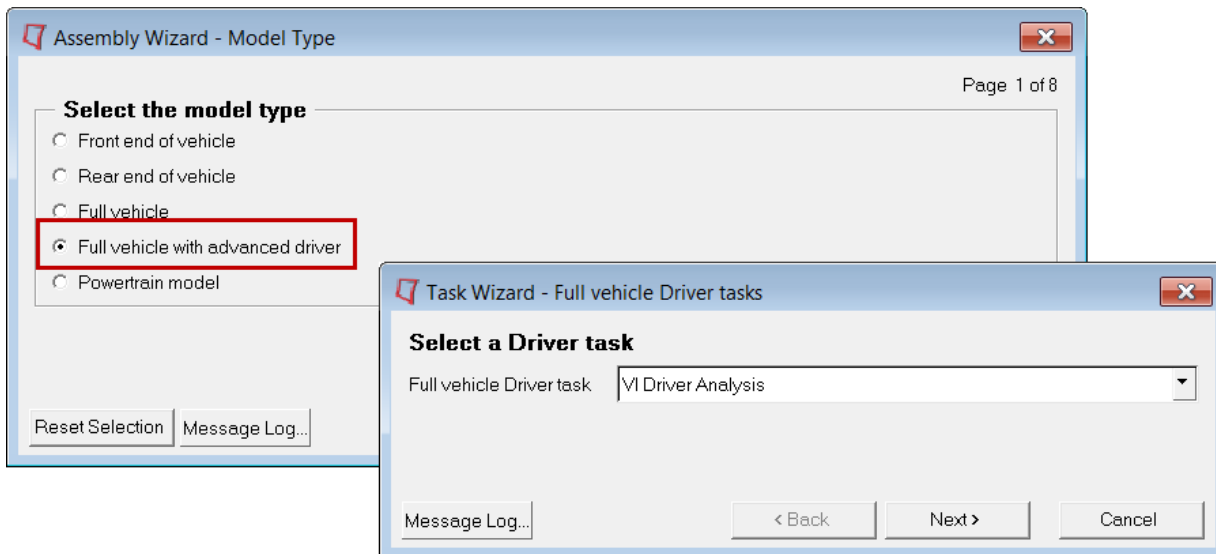
The open loop driver allows the user to hook up any signal to the steering angle, throttle, gear, clutch and brake solver variables. This allows the user to run a full vehicle using the CSE Powertrain and the disk brake sub-system



VI-Driver

MotionView 13.0 includes an environment variable driven option to run full vehicle models using VI-driver. Setting `HW_MV_EXPERIMENTAL = ADVANCED_DRIVER`, enables an option in the assembly wizard to build a full vehicle with advanced driver. After creating the model, the task wizard allows you to add a VI-Driver task and defaults to the coast event (`coast.vdf`).

This allows the user to create a Full Vehicle Model capable of running the VI-Driver developed by VI-Grade



Updated Tire Versions

Tire model versions supported by MotionSolve have been updated. The following table lists the updated versions for each type of tire models.

	Previous Version	New Version
CD Tire	3.0.3	4.0.0

COSIN Tires	2011-3	2014-2
TNO Tires	6.1	6.2.0.2

Resolved Issues

- User Tire Graphics: An incorrect argument in the grasub template was causing the tire graphics to be displayed incorrectly when using HTire. This has been fixed.
- Macpherson kingpin axis (SDF): The kingpin pin axis is now a vector through the upper strut location and the lower ball joint location instead of using just the lower ball joint, as was the case in previous releases.
- AutoSpring Preload: The set up auto-spring preload option now correctly sets preload conditions
- CDTire Auxiliary Outputs: The CDTire outputs for Tire Kinematics and Tire velocities information is now correctly provided corresponding to the requested tire.
- Open CRG Binary Files: Fixed issue where MotionSolve would error out when FTire/HTire encountered a binary OpenCRG file. Fixed issue where MotionSolve would error out when a TNO tire encountered a binary OpenCRG file.
- CDTire User Road: CDTire user road now supported via the 4.0.0 interface provided by CDTire. This can also be used via the MotionView GUI.
- CSE Powertrain – Engine Stall: Fixed issue making the CSE Powertrain less prone to stalling the engine during large changes in the Engine speed state by checking for the stall criterion only on converged steps.

Known Issues

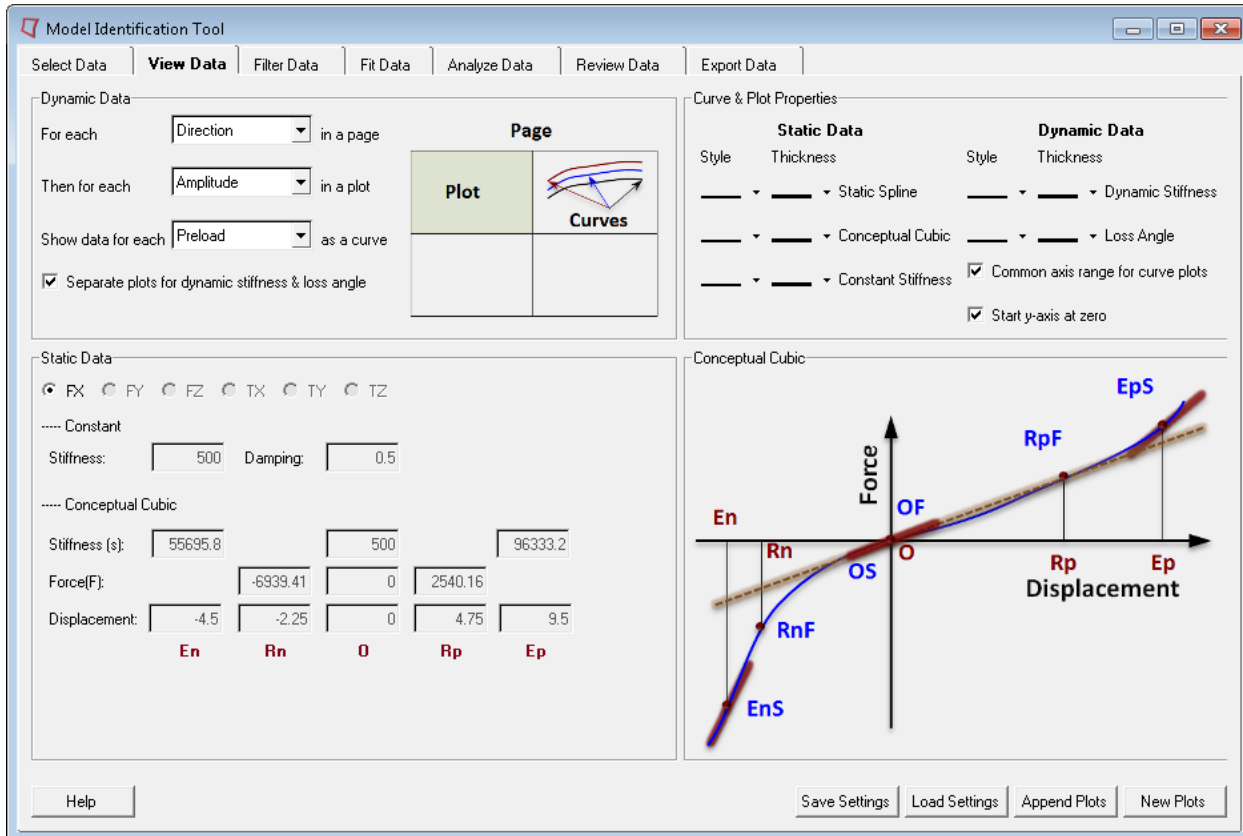
- Vehicle Dynamic Tools Auto-Entities cannot be selected from the graphics view for modification. You must select the entity in the model browser.
- Vehicle Dynamics Tools preference option is not being loaded when invoking other clients and then switching to MotionView. It is recommended to invoke MotionView directly to use the Vehicle Dynamics tools.
- ADM files with expressions having large blank spaces from the beginning of the line leads to expressions being truncated.
- An increase of memory usage and model opening time is observed with large set of assemblies sharing system definition through external files.
- "Material inside" information related to box, cylinder and sphere graphics is not saved into the mdl file.
- Browsing for file name from File fields on Linux and Mac shows full path in the file select dialog instead of file name only.
- Message related to insertion of dummy bodies for a flexible body is repeated while exporting to Adams.
- When the origin for the Points collector in the Inertia Coordinates tab of the Body panel is unresolved, the CG inertia summary assumes inertia is specified in the global coordinate system.
- F1 keyboard shortcut help on Vectors panel leads to an Error page.

The Altair Bushing Model

HyperWorks HW13.0-HWDesktop contains new features for supporting frequency, and amplitude dependent bushings. The implementation consists of four different components that work together. Collectively, they provide a comprehensive capability for identifying bushings from test data and, subsequently, using bushing models in a simulation. The components are summarized below.

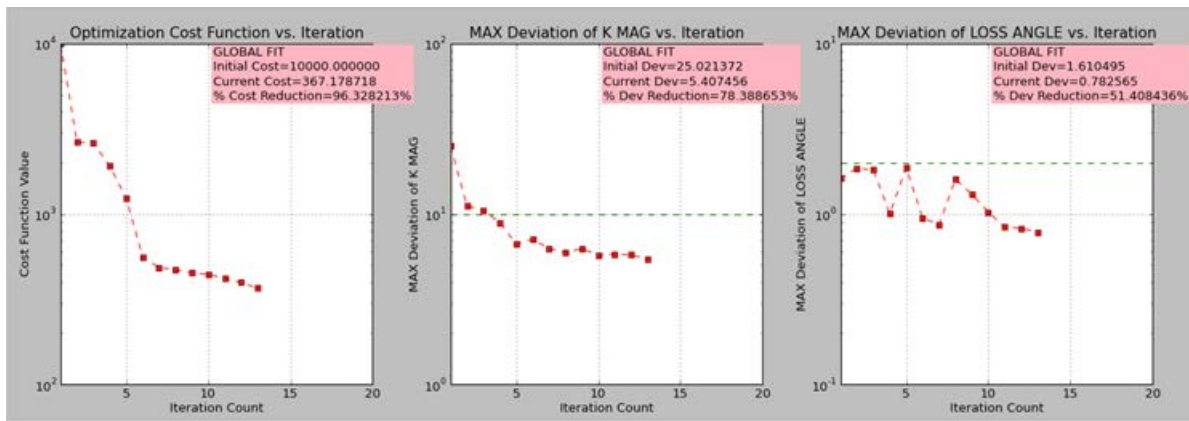
Model Identification Tool

The MIT manages the workflow associated with fitting experimentally-measured bushing data to analytical bushing models and storing these in ASCII property files. The models in this context are for frequency, amplitude and preload dependent bushings. Two dynamic and three static models are supported. The MIT is launched from HyperGraph.



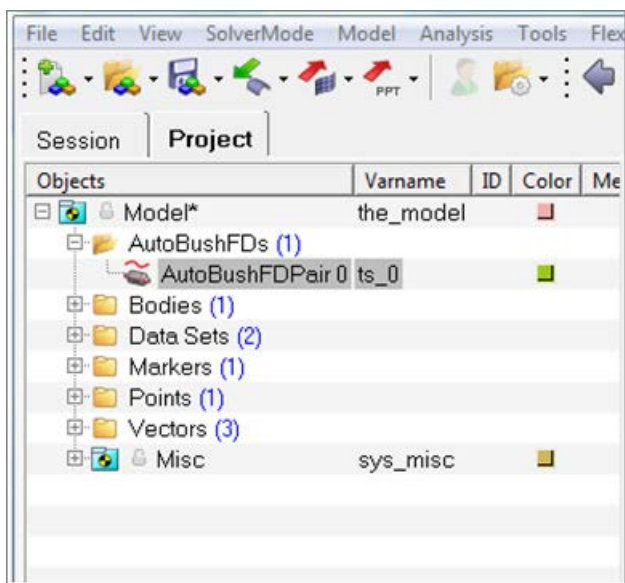
MIT Fitting Tool

This tool is invoked by the MIT user interface. It applies sophisticated numerical methods to calculate model coefficients so that deviations between model behavior and known experimental data are minimized. The fitting tool is an extension to HyperGraph.



Bushing Model in MotionView

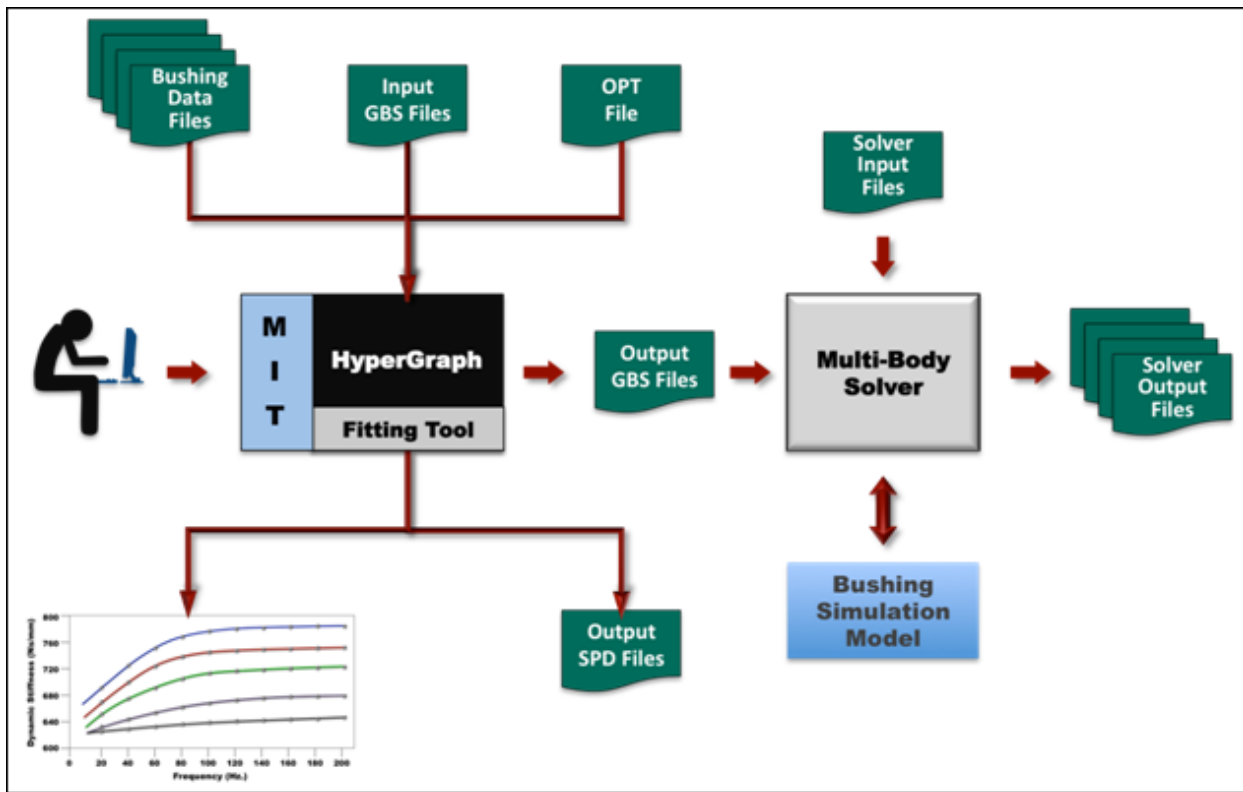
This is a set of panels in MotionView to instantiate and modify frequency, amplitude and preload dependent bushings in a vehicle model.



Altair Bushing Simulation Model

The Altair Bushing Simulation Model is used with fitted bushing models in a multibody simulation. It is a solver run-time demand load library (.dll or shared library) that works with MotionSolve. Adaptors for Adams/Solver™, MATLAB™ and SIMPACK™ are also available.

The use of this new capability is illustrated in the figure below.



The user interaction with this system is as follows:

- The user interacts with the MIT to fit bushing test data to analytical models. The output of the fitting process is a GBS file that contains the coefficients for the analytical models. The GBS is thus an analytical representation of the test data.
- A full vehicle model is built in MotionView. Bushings are used as flexible connectors at various places in the vehicle.
- The model is passed to MotionSolve, the MBD solver. At run time, MotionSolve loads the Busing Simulation Model dll (or shared library) to evaluate bushing behavior.
- The output from the solver is a set of files that describe the complete response of the vehicle for the test that was performed.

For more detailed information, please see the release notes for the [Model Identification Tool](#).

HyperCrash 13.0 Release Notes

13.0 Release Notes

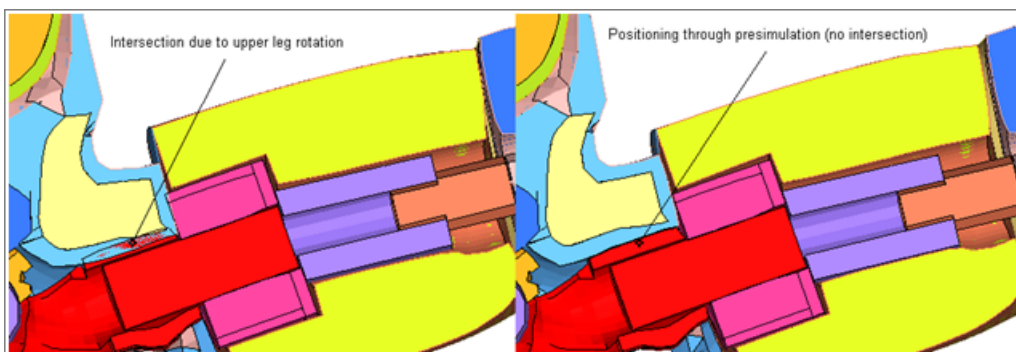
Support of Reference Geometry

It is now possible to apply the reference geometry (XREF or *INITIAL_FOAM_REFERENCE_GEOMETRY) to the model and create a reference geometry by selecting parts of the model.

A new reference geometry panel is available in RADIOSS and LS-Dyna user profile that allows listing and reviewing the initial state of the mesh.

Dummy Positioning through Pre-Simulation

Large rotation of the dummy limbs can induce intersections in the dummy model. If these intersections are not addressed they can lead to wrong kinematics and results. Dummy positioning with pre-simulation allows moving the dummy limbs without creating intersections.



Seat Deformer through Pre-Simulation

Seat foam deformation through pre-simulation is available in HyperCrash. This technique allows you to:

- Achieve better mesh quality compared to traditional morphing techniques
 - Remove intersection and most of the penetrations
 - Take into account pre-stress (through reference geometry)
-

Lock/freeze Feature

Allows locking some entities against modification, deletion, renumbering during the session or in a new session (the lock status is saved at export).

Unresolved ID Management

HyperCrash now supports unresolved IDs. This feature allows retaining the ID even if the entity referenced by this ID is not defined in the imported file. For example, importing an include file containing parts which material is not defined in the imported file but in another file.

Support of Parameters

HyperCrash now supports parameters for LS-Dyna and RADIOSS (new feature in RADIOSS v12)

Support of Groups

Groups can now be created in a separate panel in HyperCrash for the RADIOSS and LS-Dyna user profiles. These groups can then be subsequently referenced in the relevant entity (contacts, initial velocity, boundary condition...)

13.0 Release Notes - RADIOSS Block Format User Profile

RADIOSS Block 13.0 Support

A new RADIOSS user profile has been created to support RADIOSS 13.0. The supported keywords are listed below.

New Keywords

- /MAT/LAW81: Drucker-Prager with cap
 - /MAT/LAW83: Advanced connection material
 - /FAIL/TAB1: Strain failure with dependence on lode angle
 - /FAIL/NXT: Forming limit diagram based on stresses
 - /FAIL/SNCONNECT: Failure according to plastic strain criteria
 - /INICRACK: Initial crack definition (for XFEM)
 - /INSHE/THICK and /INISH3/THICK: Define initial thickness for shell elements
-

Modified Keywords

- /FAIL/CONNECT: Add ISYM for failure in compression deactivation
- /FAIL: Support optional failure ID
- /INTER/TYPE2: Support new flag Ignore=3
- /INTER/TYPE3: Support idel -1/-2 flag
- /INTER/TYPE5: Support idel -1/-2 flag
- /INTER/TYPE7: Support idel -1/-2 flag and new options related to heat generation through friction
- /INTER/TYPE10: Support idel -1/-2 flag
- /INTER/TYPE11: Support idel -1/-2 flag
- /INTER/TYPE18: Removed Istf flag and renamed Stfac to Sftval
- /INTER/TYPE19: Support idel -1/-2 flag
- /INTER/TYPE20: Support idel -1/-2 flag
- /INTER/TYPE21: New flags to support heat generation through friction
- /INTER/TYPE24: Support new card format and flags to
- /MAT/LAW10: Support new option Pext
- /MAT/LAW20: Remove density flag
- /MAT/LAW21: Support new option Pext

- /MAT/LAW36: New format allows defining more than 10 curves for strain rate dependency
- /MAT/LAW51: Support for form=11 (Drucker-Prager Yield criteria)
- /MAT/LAW76: Support new card format for strain rate dependency
- /MAT/LAW77: Support new card format for internal and external gas
- /MAT/LAW80: Support new flags Xscale_i
- /PROP/SPH: Support definition of direction of orthotropy
- /PROP/TYPE9: Support Ismstr=11
- /PROP/TYPE12: Support for new flags (fct_IDft, Yscale_F and Xscale_F)
- /PROP/TYPE14: Support Itetra=3 and Ismstr=11
- /PROP/TYPE28: Support new flags (X_SCAL and Y_SCAL)
- /PROP/TYPE43: Support new Ismstr option
- /TABLE: Support new flag for function scale

Panel Improvements

The following entities can now be created in HyperCrash:

- RBE3 panel in the Mesh Editing menu allows the creating and modifying of the RBE3 element
- Dedicated BOX panel in the RADIOSS Tools menu allows the reviewing, creating and modifying of /BOX
- Interface type1, 9 and 12 can now be created via the Contact menu
- Material type 5, 11 and 20 can now be created in the Material menu

Kinematic Joint (KJOINT2)

Added graphical representation and creation capabilities for the different type of joints (revolute, spherical, cylindrical...)

13.0 Release Notes - LS-DYNA User Profile

Include File Import

This new functionality allows the import of only some include of a complete deck by selecting the needed include from the main deck.

Replace Parts by Pairing

This functionality improves the accuracy of the replace functionality when replacing several parts.

Newly Supported Load Types

- *Boundary Temperature
- *Database Pwp Flow
- *Load Blast
- *Load Blast Segment

Part Panel Enhancement

Part type column added to display the icon corresponding to part type: rigid, initial condition, unresolved, connection.

13.0 Release Notes - Dyna2RADIOSS Converter

Enhanced Conversion

The conversion was improved for:

- Several material laws (*MAT_024; *MAT_034 ; *MAT_081 ; *MAT_089 ; *MAT_089)
- Contact parameters (friction and contact thickness options)
- LS-Dyna titles to RADIOSS titles
- *SECTION_BEAM with STYPE08 and 09
- *CONSTRAINED_JOINT

Newly Converted Keywords

- *AIRBAG_SIMPLE_PRESSURE to /MONVOL/PRES
- *AIRBAG_INTERACTION to /MONVOL/COMMU
- *CONTACT_AIRBAG_SINGLE_SURFACE to /INTER/TYPE23

12.0.115 Release Notes

Resolved Issues

- Resolved issue where HyperCrash was resetting the master node location regardless of the value of the IcoG flag
- Resolved issue where current MCO was ignored during the entity creation process if the element type used in that MCO was similar to the one used in the default MCO

12.0.113 Release Notes

General

- **Lock/Freeze feature:** Allow locking some entities or filing against modification, delete, renumbering into session or into a new session as the lock status is saved into the exported deck
- **Save View feature:** Freeze displayed screen in the session and outside session
- **Export Selection:** New "simplified export selection" and redesign and fix of classic "export selection"
- **Include setting panel:** Allow displaying min/max ID and range bounding failure in red and some automatic and manual fix to renumber all IDs

- **Modified Include File Marker:** End name of include file will include "*" sign, if an entity is created, modified, or deleted from generic panel or browser. Some other ways to modify a model need to be plugged too such as: drag and drop, renumber, node move on mesh(...?)
-

Safety Tools

- **DUMMY POSITIONER with pre-simulation:** Allow to deform intersected part of dummies
 - **NEW SEAT DEFORMER by pre-simulation:** Allow do deform seat with pre-simulation on intersected dummy/seat
-

Connectors

- **MCF Enhancements:** Key and property tags supported for bolts
 - **Part of connection:** Part of connection is now maintain during the session with a special icon and special behaviour
 - **Connection panel:** Changing a material or a property from connection panel is working now correctly
 - **Realization with existing material or property:** Material of property loaded from xml is used in realization auto or one-by-one. During replace the property or Material defined as "FIXED" into ".mco" file is maintain in the connection involved into replace part
-

RADIOSS Profile

- **Transformation Matrix Enhancements:** Full support for /TRANSFORM/MATRIX card in RADIOSS
 - **/PARAMETER support:** Full support in RADIOSS 12 profile
 - **Edit Support Enhancements:**
 - New cell support widget to be able to set the support by Id or to create simply on the fly
 - New menu to create and manage all group type : ([FORD] like group by SUBSET ...)
 - **Include File Enhancements:** Allow the reading of a deck and include in different RADIOSS version
 - **Include File Management:** Range ID of include and lock status are now exported into each individual include file
 - **Result Initialization Feature:** Allow result initialization on a selection of part from Tree or Browser
 - **LSDYNA to RADIOSS Enhancements:** Lots of converter enhancements
 - **Modularization:** Some important fixes for Modularization project and some other issues
 - **De-Penetration tool Enhancements:** Computation with "lgap = 3"
 - **Local skew frame Enhancements:** Local skew frame have now "View" options and "Mesh support" feature
-

LS-Dyna Profile

- **New Include File Enhancements:** Allow to import only some include of a complete deck by selecting the needed include from the main deck
- **New Replace Part by pairing:** Allow to select the one by one part pairing in one shot
- **New Entities "Loadcase" fully supported:** "Boundary Temperature", "Database Pwp Flow", "Load Blast", "Load Blast Segment"
- **New Entities "Tools" fully supported:** "Define Cpm...", "Define filter", "Interface Component", "Interface Linking"

- **Part panel Enhancement:** "Part type" column added to display the icon corresponding to part type : rigid, initial condition, unresolved, connection...
-

Other UI Enhancements

- **New Search by ID option:** Nice and advanced feature to search entities by id from generic panel
- **Cross reference Enhancements:** New "Select in Browser" popup feature
- **Browser open at the file loading:** Any new imported deck will now open the browser panel: nor in include hierarchy mode (if some include are detected) nor in model hierarchy mode

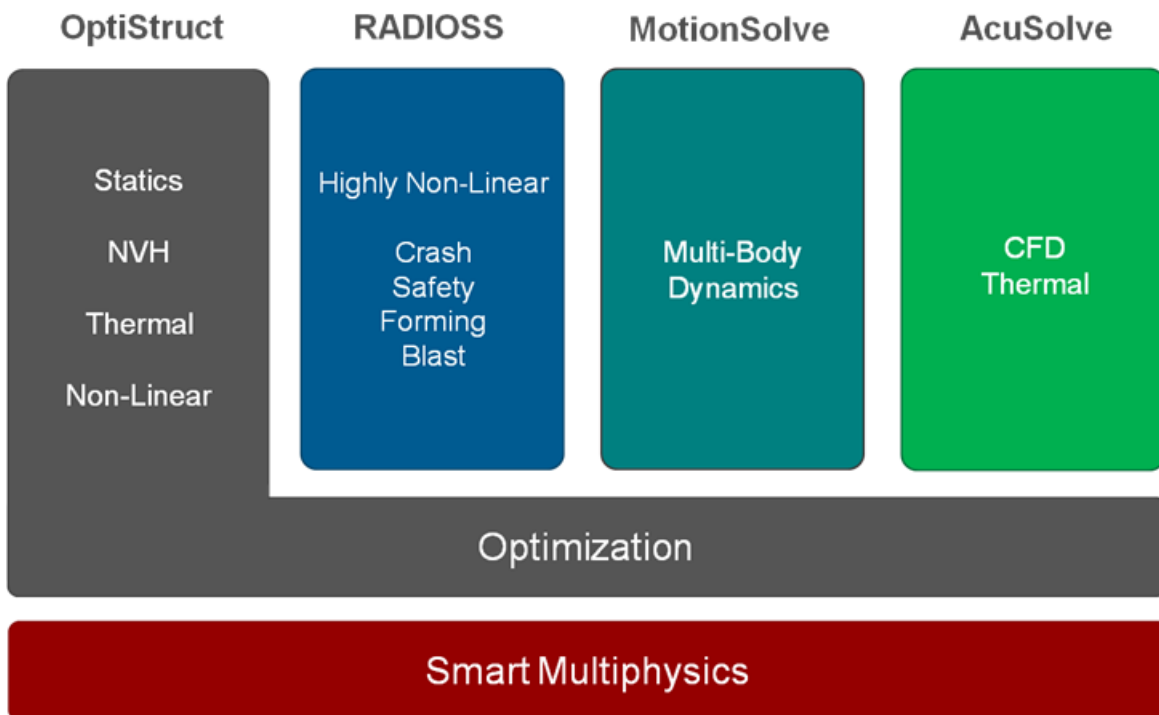
HyperWorks Solvers

HyperWorks Solvers is a collection of finite element and multibody dynamics solvers for structural, fluid-dynamics and systems simulation. These solvers can be employed for simulation driven design as well as design optimization.

OptiStruct	Design and optimization software using finite elements and multibody dynamics
RADIOSS	Finite element solver for linear and nonlinear problems
MotionSolve	Multibody dynamics solver
AcuSolve	General purpose finite element computational fluid dynamics (CFD) solver
HyperForge Solver	New solver targeted for extrusion related open die forging applications
HyperXtrude-RTM Solver	Finite element based solver for analysis and design of resin infusion processes

With the 13.0 release, a renaming of our structural solver products has taken place in order to accommodate the growing trend of applying optimization technologies more intensively in the product development process, instead of just single run analysis. Moving forward, our structural and thermal analysis and optimization product for linear and non-linear events will be called OptiStruct; this includes analysis disciplines such as linear and non-linear statics, noise and vibrations and thermal analysis. Our product for highly-nonlinear, typically transient events such as crash, impact, forming or blast analysis will be RADIOSS. Both codes are developed with a strong focus on the above mentioned disciplines, and will continue to leverage technology mutually. OptiStruct will continue to leverage the widely used bulk data input format, and RADIOSS will continue to use the well suited block format.

Moving forward, OptiStruct will also serve as the general framework for optimization involving multiple physics including CFD and FSI with AcuSolve and multi-body dynamics with MotionSolve, and will also offer API to integrate additional solvers and solutions within a general multi-model framework.



OptiStruct 13.0 Release Notes

The 13.0 release of OptiStruct is yet another functionality-rich release. This release features a variety of new and improved capabilities, including the support of poro-elastic (BIOT) material properties, Dang Van factor of safety calculations, and many more. Some key highlights of the 13.0 release are listed below:

- Large Displacement Nonlinear Static Analysis
- Hyperelastic materials
- Nonlinear Heat Transfer Analysis
- Frequency-dependent material properties in Direct and Modal Frequency Response Analysis
- Rotor dynamics for Modal Frequency Response and Complex Eigenvalue Analysis
- Parallelization using Domain Decomposition Method for reduced solution times
- Parts and instances – build hierarchical models from individual parts and multiple instances of a single part
- Subcase specific modeling – Analysis of multiple structures in a single solver run
- Global-Local Analysis – Improved solution accuracy of model substructures
- Periodic boundary conditions – Results one side of the model can be matched with the other side
- Responses can be selected for particular time-steps in MBD/NLGEOM-ESLM optimization
- External responses can be defined using Microsoft Excel workbooks
- Multi-model optimization - Run multiple optimization models with common design variables in a single run

Introduction

Altair OptiStruct is an industry proven, modern structural analysis solver for linear and non-linear structural problems under static and dynamic loadings. It is the market-leading solution for structural design and optimization. Based on finite-element and multibody dynamics technology, and through advanced analysis and optimization algorithms, OptiStruct helps designers and engineers rapidly develop innovative, lightweight and structurally efficient designs. OptiStruct is used by thousands of companies worldwide to analyze and optimize structures for their strength, durability and NVH (noise, vibration and harshness) characteristics.

Stiffness, Strength and Stability

Large Displacement Nonlinear Static Analysis

Large Displacement Nonlinear Static Analysis is now available for the solution of problems in which the load-response relationship is nonlinear and large displacements are involved. Currently, the major large displacement nonlinear capabilities include large strain elasto-plasticity, hyperelasticity of polynomial form, contact with small tangential motion, and rigid body constraints. Large Displacement Nonlinear Analysis is currently supported only for solid elements, RROD, RBAR, and RBE2 entries. All other elements will be solved using small displacement analysis.

PARAM, LGDISP can be used to activate the Large Displacement Nonlinear Static Analysis and the NLPARM subcase entry and the NLPARM bulk data entry can be used to specify the parameters of the nonlinear solution. If constraints or contacts are defined in the model, hash assembly use is recommended for Nonlinear Analysis via PARAM, HASHASSM.

Hyperelastic Materials for Large Displacement Nonlinear Static Analysis

Hyperelastic materials based on the generalized Mooney-Rivlin model (Polynomial model) are now available for Large Displacement Nonlinear Static Analysis. Hyperelastic materials can be specified using the MATHE bulk data entry and its properties can be referenced on the PLSOLID bulk data entry for CHEXA, CPENTA and CTETRA elements.

Elasto-Plastic Materials for Large Displacement Nonlinear Static Analysis

Elasto-plastic materials based on the multiplicative decomposition of the deformation gradient can now be used in Large Displacement Nonlinear Static Analysis. The MATS1 (TYPE=PLASTIC) bulk data entry can be used in conjunction with PARAM, LGDISP, 1 to activate large strain elasto-plasticity.

Nonlinear Heat Transfer Analysis

Nonlinear Heat Transfer Analysis can be used to calculate the temperature distribution in a system, in which material properties are a function of temperature. The MATT4 bulk data entry can be used to define temperature dependent thermal material properties.

Thermal Conduction Based on Gap Opening Distance

In Heat Transfer Analysis, there can be conduction across open CGAP elements based on the open gap distance. TABLEDi data is used to specify the relationship between the open gap distance and the thermal conduction coefficient. This is specified on the PGAPHT data. The solution of this contact based Heat Transfer Analysis is performed using an iterative solution of Heat Transfer and Non-linear Structural Analysis (to determine the gap status).

Thermal Conduction Based on Contact Surface Opening Distance and Closed Pressure

In Heat Transfer Analysis, there can be conduction across open contact surfaces based on the distance between the surfaces. Also, the conduction across surfaces in contact can be specified to be a function of the contact pressure. TABLEDi data is used to specify the relationship between the open surface distance and the thermal conduction coefficient as well as the contact pressure and thermal conduction coefficient. This is specified on the PCONTHT data. The solution of this contact based Heat Transfer Analysis is performed using an iterative solution of Heat Transfer and Non-linear Structural Analysis (to determine the contact status).

Nonlinear Materials can be Temperature Dependent

The TABLEST data can be used to specify nonlinear material data for different element temperatures.

Surface to Surface Contact Algorithm Implemented

The DISCRET option on the CONTACT and TIE data can be set to S2S (Surface to Surface). Surface to surface contact generally performs better than node to surface (N2S) when the surfaces are relatively flat.

Dang Van Factor of Safety for Multi-Axial Fatigue

The Dang Van factor of safety can be used for multi-axial fatigue for both analysis and optimization.

Soderberg Method for Mean Stress Correction in Stress-Life (S-N) Fatigue Analysis

The Soderberg Method for mean stress correction can now be selected for Stress-life Fatigue Analysis. The Soderberg method is slightly different from the Goodman method, the mean stress is normalized by yield stress instead of the ultimate tensile stress.

The Soderberg method can be selected on the FATPARM bulk data entry. If Type=SN then Correction=SODERBE can be used.

Fatigue Analysis has Now Been Integrated with Modal and Direct Linear Transient Solutions

Fatigue Analysis has now been integrated with Modal and Direct Linear Transient solutions. The input data is generally similar to that of Fatigue analysis on static solutions. Refer to the corresponding reference guide entries for further information.

CONTACT and TIE Enhancements

- The ADJUST option has been added to the CONTACT and TIE data to remove any initial penetration during contact. The ADJUST=AUTO option has been improved on the CONTACT and TIE data. If ADJUST=AUTO is specified, a real value equal to 5% of the average edge length on the master surface is internally assigned as the depth criterion. Additionally, the ADJUST=ALL option is now removed.
- The CLEARANCE option has been added to the CONTACT data to specify the initial gap opening irrespective of the actual distance between the two surfaces.

Bolt Pretension Forces are Now Written to the ASCII .pret File

Use the command PRETBOLT(OPTI)=YES to create the ASCII .pret file that contains the pretension force for each bolt.

Gasket Material Properties

- Gasket material properties can now be temperature-dependent. The MGASK data accepts TABLES1 data to specify temperature-dependent material properties.
- Gasket transverse shear modulus can be specified as stress/displacement or force/area data. The MGASK data GPLTYPE is used to specify if the GPL data is the stress per unit displacement (default), or force per unit area.
- Gasket tension stiffness can be specified as a stabilization coefficient or the direct tension modulus. The MGASK data EPLTYPE is used to specify if the EPL data is the tension stabilization coefficient (default), or the direct tensile modulus.

The Thermal Field from the Last Time Step of a Transient Heat Transfer Analysis can be Used for the Thermal Load

The thermal field from the last time step of a Transient Heat Transfer Analysis can be used as a static thermal load for Structural Stress Analysis. This is done by specifying the load set ID with the TSTRU data.

The Coordinate System in Which the Mass Moment of Inertia is Output can Now be Specified

PARAM, UCORD can now be used to specify the coordinate system in which the mass moment of inertia is output. It also can be used to control the point about which the mass moment of inertia is calculated.

The Grid Point Relative to Which the Center of Gravity Coordinates are Calculated and Expressed can Now be Specified

PARAM, GRDPNT can now be used to specify the grid point relative to which the center of gravity coordinates are calculated and expressed. This can be used in conjunction with PARAM, UCORD to specify about which point the mass moment of inertia is to be calculated.

Results and Output Enhancements

- Damage results are output to the .h3d and .fat files for each loading step or event. Additionally DAMAGE(SUB) can now be used to output total damage from the fatigue subcase, and DAMAGE(ALL) can be used to output damage contribution from each fatigue event, damage contribution from each static subcase and the total damage from fatigue subcases.
 - In Transient Heat Transfer Analysis, OTIME can be used to specify time steps for which analysis results should be output.
 - The modal participation factors can now be output to the PUNCH (.pch) file using PARAM, EFFMAS.
 - The Von Mises, shear, and normal stress results are available for CBAR and CBEAM elements when the .h3d file results are displayed in HyperView. The results are segregated based on the defined cross-section type specified on the PBARL and PBEAML entries.
 - Note: This result is only available for PBARL and PBEAML defined cross-sections.
 - The CONTF data allows requests for open-closed contact status, contact force, penetration, and friction. Results can be output for each subcase, load step, event or all of the above.
 - The von Mises stress threshold above which all stress results are output for a model can be specified by PARAM, STRTHR.
-

New and Enhanced Parameters

- PARAM,LGDISP: A new parameter, PARAM,LGDISP,1 is used to activate Large Displacement Nonlinear Analysis in OptiStruct.
 - PARAM,STRTHR: A new parameter, STRTHR can be used to specify the von Mises stress threshold above which the stress results are output for a model.
 - PARAM,UCORD: This new parameter can be used to specify the coordinate system in which the mass moment of inertia is output. It also can be used to control the point about which the mass moment of inertia is calculated.
 - PARAM,GRDPNT: This parameter has been enhanced and now can be used to specify the grid point relative to which the center of gravity coordinates are calculated and expressed. This can be used in conjunction with PARAM, UCORD to specify about which point the mass moment of inertia is to be calculated.
-

Enhanced Subcase Information Entries

- **TSTRU:** In addition to using the resulting thermal field from a Steady State Heat Transfer Analysis for a thermal load on a structural analysis, the thermal field from the last time step of a Transient Heat Transfer Analysis can be used for the thermal load.

- **DAMAGE:** Output types SUB, LOAD, EVENT, or ALL can be specified on the DAMAGE I/O option entry.
- **MATFAT:** Continuation data for factor of Safety Analysis added.
- **CONTF:** The contact output can requested for forces (FORCE), pressure, open-closed status, gap opening, and penetration (PCONT), tangential friction force (FRICT), or all available contact results (ALL).
- **PRETBOLT:** The OPTI option has been added to generate an ASCII file containing the bolt pretension forces.

New Bulk Data Entries

- **PGAPHT:** Used to specify the conduction coefficients for the open and closed gap as well as a tabled relationship between the gap opening distance and the thermal conduction coefficient.
- **PCONTHT:** Used to specify the conduction coefficients for the open and closed contact as well as a tabled relationship between the contact opening distance and the thermal conduction coefficient. Also, a tabled relationship between the contact pressure and the thermal conduction coefficient can be specified.
- **TABLEST:** Used to specify different non-linear material TABLES1 data for different element temperatures.
- **MATHE:** Defines the material properties for a new nonlinear hyperelastic material based on the Mooney-Rivlin model (polynomial model).
- **PLSOLID:** Used to define the properties of a nonlinear hyperelastic solid element referenced by CHEXA, CPENTA and CTETRA elements. The MATHE hyperelastic material can be referenced to define the corresponding material properties.
- **MATT4:** The MATT4 bulk data entry is a new temperature dependent material property that can be used to define conductivity as a function of temperature in Nonlinear Heat Transfer Analysis.

Enhanced Bulk Data Entries

- CONTACT and TIE:
 - The ADJUST option has been added to remove initial penetrations.
 - The DISCRET option has been added to specify surface to surface contact.
- CONTACT: The CLEARANCE option has been added to specify the initial gap opening.
- MGASK:
 - The MGASK data now accepts TABLES1 data to specify temperature dependent material properties.
 - The GPLTYPE option can be used to specify if the GPL data is the stress per unit displacement (default), or force per unit area.
 - The EPLTYPE option can be used to specify if the EPL data is the tension stabilization coefficient (default), or the direct tensile modulus.
- FATPARAM:
 - The FATPARAM data has been enhanced with support for the Soderberg mean stress correction method. If Type=SN, then Correction=SODERBE can now be selected to activate the Soderberg method.
 - Continuation data for factor of safety analysis is now available. The FOSType field can now be set to DANGVAN for Factor of Safety (FOS) analysis.
- DRESP1: The Dang Van Factor of Safety output can now be used as a response by selecting RTYPE=FATIGUE and ATTA=FOS

Noise and Vibrations

Material Properties can be Frequency-Dependent in Direct and Modal Frequency Response Analysis

The new MATF_x data can be used to specify tables that contain the frequency-dependence of various material properties. At each loading frequency, the appropriate material properties are obtained from these tables for both Direct and Modal Frequency Response Analysis.

Rotor Dynamics

Rotor dynamics is now available for Modal Frequency Response and Complex Eigenvalue Analyses. Rotor dynamics is the analysis of structures containing rotating components. The dynamic behavior of such structures is influenced by the type and angular velocity of the rotating components and their locations within the model. Synchronous (RGYRO=SYNC) and Asynchronous (RGYRO=ASYNC) analyses are available for all solution sequences supporting Rotor dynamics.

Output data required for the Campbell Diagram can be generated using an asynchronous Complex Eigenvalue Analysis (RGYRO=ASYNC and RSPEED).

Supported input data in the Solution Control section is K2PP, M2PP, B2PP, RGRYO, SDAMP, DLOAD, FREQ, and CMETHOD. Supported PARAM are GYROAVG, WR3, WR4, G, ALPHA1, and ALPHA2. Supported bulk data are RGYRO, ROTORG, RSPINR, UNBALNC, and RSPEED.

BIOT Materials can be Used for Noise and Vibration Analysis

The poroelastic (BIOT) materials can be used to accurately capture the frequency-dependent behavior for poroelastic materials using the MATPE1 entry for Noise and Vibration Analysis.

Fluid-Structure Coupling Improved

The Fluid-Structure coupling specified by the ACMODL data has been improved to only couple with the top layer of structural GRID. In the past, all structural GRID in the search box were coupled to the fluid mesh. PARAM, ACMODL12, YES can be used to restore the ACMODL formulation used prior to OptiStruct 13.0.

Fast Parallel Solver for Large Modal Frequency Response Problems

A new parallel algorithm has been developed to quickly solve the modal frequency response equations when thousands of modes are used to define the modal space. With nearly linear parallel speed up the solver can handle problems with any combination of modal damping, viscous damping, low and medium rank structural damping in both the fluid and structure. In addition, SPCD enforced motion can be used and modal energy calculation is available. The use of this new method is controlled by PARAM FASTFR.

The DMIGMOD Data can Now be Used to Translate and Rotate CMS Super Elements

In previous versions, the DMIGMOD data could only be used to translate CMS super elements. No rotation was allowed. This restriction has been removed. The translation and rotation is accomplished by specifying three grids on the CMS super element and three grids in the residual structure that will match up with the moved CMS super element.

Enforced Motion can be Used in Modal Transient Analysis

In addition to force loads, enforced motion can now be applied in Modal Transient Analysis.

AMSES Enhancements

- Multiple Eigenvalue Analysis subcases are now allowed with AMSES. However, only one can be used for Modal Frequency Response or Modal Transient Analysis.
 - There can be multiple subcases with different MPC sets for Eigenvalue Analysis with AMSES. However, only one can be used for Modal Frequency Response or Modal Transient Analysis.
 - Prestressed Eigenvalue Analysis is now supported for AMSES runs. STATSUB(PRELOAD) can be used to reference a static subcase that defines a preload in AMSES runs.
 - The calculation time for Eigenvalue Analysis with the AMSES solver has been reduced. This is especially apparent when a large number of eigenvector DOF must be calculated.
 - AMSES can handle much larger problems. In the past, the EIGRA eigenvalue solution could not handle certain very large problems (millions of DOF and over 10,000 modes). This limitation has been greatly reduced.
 - AMSES can be used to solve for the eigenvalues of the fluid mesh. Both Lanczos and AMSES can be used to solve for the eigenvalues and eigenvectors of the fluid mesh.
-

Modal Damping Included in Complex Eigenvalue Analysis

The modal damping (SDAMPING) is now included in the calculation of complex eigenvalues.

Viscous Damping Based on Scaled Mass and Stiffness in Complex Eigenvalue Analysis

The **PARAM ALPHA1** and **ALPHA2** are now used to create viscous damping based on the scaled mass and stiffness in the calculation of complex eigenvalues.

Mode Selection

- The MODESELECT data can be used to select a subset of the normal modes to be used in Modal Frequency Response, Modal Transient, and Complex Eigenvalue Analysis.
 - The PARAM LFREQFL and HFREQFL can be used to select a subset of the fluid normal modes to be used in Modal Acoustic Frequency Response.
 - The parameters PARAM LFREQ and PARAM, HFREQ can be used to select a subset of the structural normal modes to be used in Modal Frequency Response and Modal Transient Analysis.
-

Frequency Range Selection for Response Spectrum Analysis

The parameters PARAM LFREQ and PARAM, HFREQ can be used to select the lower and upper bounds, respectively in Response Spectrum Analysis.

File to Allow Optimization of CMS Super Elements can be Generated

Using the DMIGDV continuation of the CMSMETH data, the contents of an include file used for optimization of the super element modal frequencies and their corresponding structural viscous damping values are specified.

Sets of RIGID Elements can Now be Specified with the Bulk Data SET Data

The set TYPE can now be specified as RIGID in the bulk data SET data. This is required when specifying the set of RIGID elements attached to the connection points in the PFPATH data.

The Creation Time of CDS Super Elements has Been Significantly Reduced

A new algorithm for CDSMETH super element generation dramatically reduces the calculation time.

Modal Super Elements can be Created for Use in RecurDyn by FunctionBay

The new parameter PARAM,RFIOUT is used to turn on creation of the .rfi file that contains modal super elements for use in multibody dynamics with the RecurDyn software from FunctionBay.

Interior GRID Results Recovery for CDS Super Elements

Use the OSET field on the CDSMETH data to specify the set of interior GRID for the super element in which results will be recovered in the residual run.

Results and Output Enhancements

- Sound power and sound intensity results can now be output only for microphone locations in Radiated Sound Analysis. SINTENS(PANEL/NOPANEL) and SPOWER(PANEL/NOPANEL) can be used to select the corresponding output.
 - Corner force results for shell elements can be written to the .h3d, .op2, and .pch files for Static Analysis.
 - The corner stress results for solid elements can be written to the .h3d, .op2, and .pch files for Normal Modes Analysis.
 - Previously, OLOAD only generated output for DAREA loading. Now it also generates applied load results for static loads referenced by the RLOADx and TLOADx data.
 - The corner stress results in shell and solid elements can be written to the .h3d, .op2, and .pch files for Transient Analysis.
 - Since multiple RANDOM data are allowed, the RANDOM data ID is now written to the XYPUNCH header for PSDF output. This is so the output related to each RANDOM data can be identified. Note that if there is only a single RANDOM datum, the ID will not be written to the header.
 - Output from Complex Eigenvalue Analysis can be reduced by only outputting the unstable modes. Instead of outputting all modes from a Complex Eigenvalue Analysis, the command DISP(UNSTABLE)=ALL can be used to specify only the output of the unstable modes.
 - The run time and disk space usage has been decreased when GPFORCE and MPCFORCE output is requested for Frequency Response and Transient Analysis. The reduction is quite dramatic for large problems.
-

New Solution Control Entries

- RGYRO: Identifies a RGYRO bulk data entry that contains information required to implement rotor dynamics in Modal Complex Eigenvalue Analysis and/or Modal Frequency Response Analysis.
- MODESELECT: Used in the I/O options or the subcase information section to select a subset of computed modes in Modal Dynamic Analysis Subcases.

New Parameter Data

- PARAM,FASTFR: The parameter FASTFR is used to turn on or off the use of the new parallel fast equation solver for Large Modal Frequency Response Analysis. The default value is AUTO. When set to AUTO, OptiStruct automatically determines if this solution algorithm will reduce the run time.
- PARAM,RFIOOUT: When this PARAM is set to YES during a CMSMETH run using the CBN method, a file is created with the extension .rfi. This file contains the modal super element to be used with the RecurDyn Multibody Dynamics software from FunctionBay.
- PARAM,LFREQFL: This parameter is used to exclude fluid modes in Modal Acoustic Frequency Response Analysis. Fluid modes with frequencies less than LFREQFL are not used in Acoustic Analysis.
- PARAM,HFREQFL: This parameter is used to exclude fluid modes in Modal Acoustic Frequency Response Analysis. Fluid modes with frequencies greater than HFREQFL are not used in Acoustic Analysis.
- PARAM,LFREQ: This parameter is used to exclude structural modes in Modal Frequency Response and Modal Transient Analysis. Modes with frequencies less than LFREQ are not included, and this parameter can be used to remove rigid body modes from the modal space. LFREQ is also used in Response Spectrum Analysis to specify the lower bound of the frequency range of interest for modal combination.
- PARAM,HFREQ: This parameter is used to exclude structural modes in Modal Frequency Response and Modal Transient Analysis. Modes with frequencies higher than HFREQ are not included, and this parameter can be used in Response Spectrum Analysis to specify the upper bound of the frequency range of interest for modal combination.
- PARAM,WR3: This parameter is used to include or exclude global frequency dependent damping in Rotor Dynamics Analysis.
- PARAM,WR4: This parameter is used to include or exclude material frequency dependent damping in Rotor Dynamics Analysis.
- PARAM,GYROAVG: This parameter is used to select the Frequency Response Analysis formulation type for Rotor Dynamics Analysis.
- PARAM,ACMODL12: This parameter can be used to restore the ACMODL formulation used prior to OptiStruct 13.0

Enhanced I/O Options and Subcase Information Entries

- DISP: The UNSTABLE option specifies that only the unstable mode shapes are output from Complex Eigenvalue Analysis.
- FORCE: The location option can be used to request element force results. Location=BILIN can now be used to request corner forces for shell elements.
- SINTENS: NOPANEL option can now be used to output sound intensity results for microphone grids only. The PANEL option can be used to request sound intensity results for both panel grids and microphone grids.
- SPOWER: NOPANEL option can now be used to output sound power results for microphone grids only. The PANEL option can be used to request sound power results for both panel grids and microphone grids.

New Bulk Data Entries

- MATF1, MATF2, MATF3, MATF8, MATF9, and MATF10: Material properties can now be made frequency dependent using the MATFx data that reference TABLEDx that contains the material property vs. frequency data.

- RGYRO: Identifies the type of Rotor Dynamics Analysis, the rotor, and the rotational speed.
- ROTORG: Specifies the list of grids that determine the rotor line model.
- RSPEED: Used to specify a set of reference rotor speed values for asynchronous analysis in rotor dynamics.
- RSPINR: Defines the relative spin rates between rotors during a Rotor Dynamics Analysis in modal complex eigenvalue or frequency response solution sequences.
- UNBALNC: Defines the unbalanced rotating load during a Rotor Dynamics Analysis in frequency response solution sequences. The unbalanced load is specified in a cylindrical system where the rotor rotation axis is the Z-axis.
- MATPE1: Defines the material properties for poro-elastic (BIOT) materials.

Enhanced Bulk Data Entries

- CMSMETH: Using the DMIGDV continuation of the CMSMETH data, the contents of an include file used for optimization of the super element modal frequencies and their corresponding structural viscous damping values is specified.
- SET: The SET data can now be used to specify a SET of rigid elements. This is used for One-step Transfer Path Analysis by specifying this SET on the PFPATH data.
- CDSMETH: The OSET field can be used to specify the set of interior GRID for the super element in which results will be recovered in the residual run.
- DMIGMOD: The DMIGMOD data now has specifications for three grids on the super elements and the three grids that they will match up with in the residual structure when the super element is moved into position.

Kinematics and Dynamics

Generation of Solver and Results Recovery .h3d Files

PARAM, MBDREC creates two flexible body files during the CMS generation.

A small file *.h3d contains the minimum amount of information needed for MBD simulation in MotionSolve.

A larger file *_recov.h3d will also be generated that contains the information from the small file as well as recovery information for displacement, velocity, acceleration, stress, and strain.

The large file is used along with the MotionSolve calculated result (.mrf) file specified with the ASSIGN,MBDINP data in an OptiStruct Transient Analysis to recover displacement, velocity, acceleration, stress, and strain results of the flex body. These results can be output in the .op2 and .h3d files.

Recovery of MBD Analysis Results in OptiStruct

After MotionSolve is run, it is possible to recover displacements, velocities, accelerations, stresses, and strains for the flexbody in OptiStruct in order to create .op2 and .h3d results files for Fatigue Analysis. The procedure is explained below.

After running MotionSolve, a residual run can be made to recover displacement, velocity, acceleration, stress, and strain results for interior grids and elements in the flexbody based on the modal participation results from MotionSolve. After running MotionSolve, a resulting filename.mrf file is created that contains MotionSolve results including the modal participation factors at each time step for the flexbody for Transient Analysis time step. In the residual run, the flexbody filename_recov.h3d file and the .mrf results file are specified using the ASSIGN data:

- ASSIGN,H3DMBD,30101,'pflexbody_1_recov.h3d'

- ASSIGN,H3DMBD,30102,'pfbbody_2_recov.h3d'
- ASSIGN,MBDINP,10,'pfbbody.mrf'

Where the *10* in the ASSIGN,MBDINP data corresponds to the SUBCASE for which the MotionSolve results will be used. In SUBCASE 10, instead of performing a Transient Analysis, OptiStruct will just use the results from MotionSolve.

The *30101* and *30102* in the ASSIGN,H3DMBD data refer to the flexbody's ID in the .mrf file.

Preload can Now be Accounted for During Flexbody Generation

The effect of preload on the flexbody is taken into account as the flexbody matrices are calculated when the LOAD LID and SPC SID are specified on the CMSMETH data.

DISP(H3D)=NONE Works When Creating a Flex Body for ADAMS

The use of DISP(H3D)=NONE will reduce the size of the .h3d file when OUTPUT,ADAMSMNF,YES is specified.

Residual Vectors can be Used to Increase the Accuracy of Flexbodies

Residual vectors based on the LOAD LID specified on the LOADSET continuation of the CMSMETH data are used to increase the accuracy of results for the flexbody.

Analysis Results can be Generated Using the Results from SIMPACK Analysis

After running a Multibody Dynamic Analysis in SIMPACK, the resulting modal participation coefficients in the .unv file can be used by OptiStruct to recover the dynamic displacements, velocities, accelerations, stresses, and strains. The ASSIGN statement keyword SIMINP is used to specify the .unv file.

Element Forces for CMSPDPC Output

The element forces for the CMSPDPC elements are now written out in the results file with the MBFORCE request.

Output the .exb file for AVL/EXCITE Directly from OptiStruct

The parameter PARAM, EXCEXB can be used to control the output of the AVL/EXCITE .exb file directly from OptiStruct.

Enhanced Solution Control Data

- ASSIGN,SIMPINP: Used to specify the .unv file that contains the CMS flexbody modal participation coefficients generated from a SIMPACK Multibody Dynamic Analysis.
- ASSIGN,MBDINP: Used to specify the .mrf file that contains the CMS flexbody modal participation coefficients generated from a MotionSolve Multibody Dynamic Analysis.
- ASSIGN,H3DMBD: Used to specify the .h3d file that contains the CMS stress, strain, displacement, velocity, and acceleration model results vectors to be used to calculate results using the .mrf file generated by MotionSolve.

Enhanced Bulk Data Entries

- CMSMETH: A LOAD LID and SPC SID can be specified on the PRELOAD continuation to preload the flexbody. The effect of the preload on the flexbody is taken into account when the flexbody matrices are calculated.
- CMSMETH: A LOAD LID can be specified on the LOADSET continuation to generate residual vectors that can increase the accuracy of the flexbody results. The degrees of freedom of the loads are used to create residual vectors.

New Parameters

- PARAM, MBDREC: creates two flexible body files during the CMS generation.
 - A small file *.h3d contains the minimum amount of information need for MBD simulation in MotionSolve.
 - A larger file *_recov.h3d will also be generated that contains the information from the small file as well as recovery information for displacement, velocity, acceleration, stress, and strain.
- PARAM, EXCEXB: Controls the output of the AVL/EXCITE .exb file directly from OptiStruct.

General

Build Hierarchical Models from Different Individual Parts and Include Multiple Instances of a Single Part

The new Parts and Instances capability allows you to build hierarchical models from existing input decks without entity renumbering requirements (preserving original grid or element ID's). This feature allows for greater flexibility in the creation of a finite element model. Independent substructures can be defined which are easily combined into a final larger model using simple translational and rotational transformations.

It is possible to duplicate parts (thereby creating multiple instances of a part) by including the same file multiple times and then these parts can be relocated to the required positions. A majority of the input remains in the traditional format; however some entries can be defined in a generalized format, to be used as connectors, because they can contain explicit references to GRIDs in different parts. Such connectors include: CBUSH, CELAS, CWELD, CONTACT, MPC, all RIGID elements. Parts can also be connected automatically using matching GRIDs. SPCADD, LOADADD, MPCADD are generalized to allow data collection from various parts. Properties and materials are shared between different parts using the standard format.

The new bulk data entries INSTNCE, RELOC, and CONNECT are used to position, define relocation, and connect various parts in the model, respectively.

In OptiStruct 13.0 this functionality is limited to analysis runs. Nonlinear Transient Dynamics via RADIOSS integration, Multibody Dynamics, SPMD parallelization, and Optimization runs are not currently supported. This functionality is not currently available in HyperMesh; connectors and assembly specific entries need to be manually edited in the OptiStruct input file.

Analyze Multiple Structures in a Single Solver Run using Subcase Specific Modeling

Multiple structures can now be analyzed in a single solver run using the new Subcase Specific Modeling capability. The structures can be completely independent; can represent different regions of the same model, or different assemblies sharing common parts. The solution time may be reduced for situations involving the solution of models with small interchangeable sections. This capability can also be used to activate/deactivate specific elements between subcases (element birth or death).

Each subcase can now be assigned a different SET of elements representing a sub-model. There is no requirement for any relation between such sub-models. Subcase specific modeling is activated by the use of the SUBMODEL subcase entry. This SUBMODEL entry selects an element set (and optionally a set of rigids) that defines the submodel to be analyzed within that subcase. This functionality is available for Linear Static Analysis only. All optimization types with responses from Linear Static Analysis are supported (except SPCFORCE/residual force responses).

Improve Solution Accuracy of Model Sections using Global-Local Analysis

Global-local Analysis (sometimes called Zooming) is implemented using the Subcase Specific Modeling functionality. Some subcases in this approach can represent a coarse model of the full structure (global subcase), and other subcases can represent detailed models of a subset of the full structure (local subcase). A transfer zone has to be designated by a set of grids or elements, and OptiStruct automatically approximates the solution from the selected elements in the global subcase to the boundary grids in the local submodel. The GLOBSUB subcase entry is used to select the global structure subcase and the set of grid points of the local structure that define the transfer zone. The transfer zone is limited to volume elements in both subcases, and this formulation is available for all solution types supported for subcase specific modeling. For optimization runs, any design variables affecting the local submodel should also be mapped to the global submodel.

This functionality can possibly be used to gain higher solution accuracy in sections of high stress concentrations or in regions of interest where higher accuracy is required.

Infinite or Cylindrical Periodic Structures can Now be Solved Using Periodic Boundary Conditions

A set of grids on one side of a model can now be offset to the other side. OptiStruct will automatically match the solution of the model at such grids (with similar locations). Infinite periodic structures with periodic loads and supports can now be solved using this functionality. The Periodic Boundary Condition functionality is only available in linear static analysis for OptiStruct 13.0. All optimization types with responses from Linear Static Analysis are supported (except SPCFORCE/residual force responses).

The PERBC bulk data entry is used to define a connection between opposite edges/faces of the structure. Opposite sides can be of arbitrary shape (flat or curved) but require matching location of grids on both sides of the model. Multiple pairs of matching sides are allowed (periodicity in two or three directions). This functionality is not currently supported by HyperMesh, however, it can be directly edited in the OptiStruct input file.

Domain Decomposition Method (DDM) is Now Implemented to Reduce Solution Time

The Domain Decomposition Method (DDM) is a new parallelization approach to reduce solution times for linear static, nonlinear static analyses and optimization. The DDM method utilizes a graph partition algorithm to automatically partition the geometric structure into multiple domains (equal to the number of MPI nodes). The analyses are then carried out in the individual domains and finally communicated across domains to generate the final solution. The DDM method can be activated using the `-ddm` run option.

Von Mises Stress and Strain Results for Random Response Analysis Based on the Segalman Method

The von Mises stresses and strains can now be output for Random Response Analysis based on the Segalman Method. The existing PSDF and RMS options on the STRESS I/O options entry now includes von Mises stresses and strains for Random Response Analysis subcases.

Mumps Support

Mumps can now be used on Macintosh computers.

The Maximum Amount of Memory to be Used can be Specified

By default, OptiStruct uses the minimum amount of memory required. The new .cfg file option USERAM specifies the maximum amount of memory to be used. The memory will be increased up to this value if the solution time will be reduced due to more memory.

The Solver GUI Now Generates Plots as the Job Runs

Plots of optimization objective function and maximum constraint violation can be generated as the solution progresses.

AMLS Support

- Version 5 of AMLS can be used with OptiStruct version 13.0.
- For version 5 of AMLS, the memory used by AMLS can be set using the run option `-amlsmem`, the parameter `PARAM,AMLSMEM`, or using the environment variable `AMLS_MEM`. The run option overrides the parameter and the environment variable. If both the parameter and the environment variable are set in a model, the environment variable `AMLS_MEM` will override the value set by the parameter.

FastFRS Support

- Version 2 of FastFRS can now be used with OptiStruct version 13.0.
- FastFRS version 2 can now be used with OptiStruct and the memory used by FastFRS (version 2) can be set using the run option `-ffrsmem`, the parameter `PARAM,FFRSMEM`, or using the environment variable `FFRS_MEM`. The run option overrides the parameter and the environment variable. If both the parameter and the environment variable are set in a model, the environment variable `FFRS_MEM` will override the value set by the parameter.
- FastFRS can now be run using the `-ffrs` run option. Additionally, the `-ffrsncpu` run option can be used to specify the number of CPU's used for the run.

Overly Stiff Bushings and Springs can be Automatically Set to a Reasonable Value

The parameters `BUSHSTIF` and `ELASSTIF` are used to specify upper bounds of translational and rotational stiffnesses for `CBUSH` and `CELAS` elements. Any corresponding stiffness value above the value specified by the parameter will automatically be reset to the value of the parameter. Additionally, the four parameters `ELASTLMT`, `ELASRLMT`, `BUSTLMT`, and `BUSHRLMT` can be used for stiffness control by specifying limiting values. If the corresponding translational or rotational stiffness values are greater than the limits, then `WARNING` messages are output.

Rigid Elements with Double Dependency Issues

To avoid double dependency issues with rigid elements, their dependent degrees-of-freedom may now be converted to independent degrees-of-freedom when conversion is necessary for the model to run. This conversion can be used if the dependent degree-of-freedom of a rigid element is `SPC`'ed, part of the interface degrees-of-freedom (specified using `ASET`, `CSET`, `BNDFREE`, and so on), or if it is a dependent degree-of-freedom of another constraint equation (an `MPC` equation or another rigid element). This conversion from dependent to independent degrees-of-freedom is done automatically when `PARAM,AUTOMSET,YES` is specified in the input data.

ORDERING Method for MUMPS Solver can be Specified

The ordering method (AMD, ORD, METIS, or AUTO) can be specified on the SOLVTYP data for the MUMPS solver.

A New "Light" Version of the Nonlinear Expert System can Now be Activated

PARAM, EXPERTNL, AUTO can now be used to activate a new "light" version of the expert system, which is designed to facilitate convergence of a nonlinear process in a reasonably close to minimum number of iterations. This version may adjust the time step, including increasing the time step beyond that prescribed on the NLPARM entry.

Thin Bending Shell Elements can Now be Automatically Converted to Membrane Elements

Thin shell elements (thickness 0.001 or less) can cause numerical issues such as singularities when bending material properties are specified using MID2 and MID3. These elements are generally used to skin solid elements for surface stress recovery purposes. They should be converted to membrane elements by removing MID2 and MID3. This can now be done automatically using PARAM,SHL2MEM,value, where value is the upper bound thickness at which the bending elements will automatically be converted to membrane elements.

CBAR and CBEAM can Optionally Reference PBEAM and PBAR Data, Respectively

In the past, CBAR data could reference PBEAM data and CBEAM could reference PBAR data. Whether or not this is allowed is now controlled by the SYSSETTING command BARPROP. If BARPROP is set to STRICT (the default), then, CBAR and CBEAM cannot reference PBEAM and PBAR data, respectively. If BARPROP is set to MIXED, then CBAR and CBEAM can reference PBEAM and PBAR data, respectively.

Results and Output Enhancements

- The STRAIN(MODAL) request will generate STRAIN output for the structural and RESVEC modes to the .op2 and .pch files for frequency response and transient analyses.
- GPFORCE for normal modes when USET,U6 data is present is now written to the .op2 and .pch files. In addition, for Modal Frequency Response and Modal Transient Analysis, the GPFORCE for RESVEC are written to the .op2 and .pch files when the GPFORCE(MODAL) command is used.
- MPCFORCE for normal modes when USET,U6 data is present is now written to the .op2 and .pch files. In addition, for Modal Frequency Response and Modal Transient Analysis, the MPCFORCE for RESVEC are written to the .op2 and .pch files when the MPCFORCE(MODAL) command is used.
- The new parameter RECOVER can be used to output the full-structure modes, within a range of frequencies specified by LB (lower bound) and UB (upper bound), during component mode synthesis (CMS) instead of the modes of the condensed system.
- The new parameter RECOVER only creates results for the .op2 file, and an error will be issued if no .op2 file results are requested.
- CFAST element forces are now included in the .op2 file.
- Element forces for CBAR, CBEAM, CROD, CBUSH, CGAP, CWELD, CFAST, CVISC, CDAMP, and CELAS are now written to the .h3d file as vector and tensor quantities as appropriate. All analysis types are covered. There are two limitations currently; HyperView 12.0.110 must be used and the .fem file must be used as the model file.
- PARAM, PRGPST can now be used to control the amount of AUTOSPC output. Valid options for PARAM,PRGPST are now YES, NO, ALL, and N, where N is the number of AUTOSPC DOF to be output. In the past, only the first 100 DOF could be printed.

- GEOM3 and GEOM4 data blocks output can be controlled by using PARAM, OP2GM34 in conjunction with PARAM, POST, -1. If PARAM, OP2GM34, TRUE and PARAM, POST, -1 are specified, then GEOM3 and GEOM4 data blocks are output to the .op2 file. If PARAM, OP2GM34, FALSE and PARAM, POST, -1 are specified, then the data blocks are not output to the .op2 file.
- When PARAM,POST,-2 is used, the GEOM3, GPDT and GPL data blocks are now included in the .op2 file. These data blocks are required by some in-house post-processors.
- In the past, THICKNESS could only be output to the .h3d file for optimization. Now it is available for analysis also.
- By default, the inertia relief rigid body forces and accelerations are no longer printed to the .out file. To turn on this printing, set PARAM,PRINFACC to 1.
- GAP element forces can now be plotted in HyperView. Note that the model file must be the .fem file and not the .h3d file, because the h3d file does not contain information about the gap coordinate system.

PARAM,RBE3FREE can be Used to Disable ERROR 772 Due to Free Spiders on RBE3 Elements

Free spiders on RBE3 elements will be singular in some cases. When AMSES or AMLS is used, this modeling error will lead to invalid results as the free spider will be AUTOSPC'ed. These results will be invalid because the RBE3 is now constrained due to the AUTOSPC. For Lanczos and Static Analysis, the singularity will cause errors 155 or 153 to be issued. PARAM, RBE3FREE,YES will change ERROR 772 into WARNING 824. If the free spider is singular, the run will fail with ERROR 155 or 153 for Static Analysis and Lanczos eigenvalue solutions. For AMSES and AMLS solutions, the AUTOSPC output should be carefully checked to make sure the RBE3 has not been constrained.

PARAM,RBE2FREE can be Used to Disable ERROR 725 Due to Rigid Body Rotation of RBE2 Elements

RBE2 elements with rigid body rotations are singular. When AMSES or AMLS is used, the free rotational DOF will be AUTOSPC'ed. For Lanczos and Static Analysis, the singularity will cause errors 155 or 153 to be issued. PARAM, RBE2FREE,YES will change ERROR 725 into WARNING 825. In order to solve the problem with Lanczos or Static Analysis, the user must use AMSES and note the RBE2 DOF that are AUTOSPC'ed, and then manually SPC these DOF in the subsequent Static Analysis or Lanczos Eigenvalue Analysis.

New Parameter Data

- PARAM,PRINFACC: The new parameter PRINFACC controls the printing of the inertia relief rigid body forces and accelerations to the .out file. By default, the inertia relief rigid body forces and accelerations are no longer printed to the .out file.
- PARAM,AUTOMSET: When the new parameter AUTOMSET is set to YES, any rigid elements with double dependency issues are fixed by the conversion of dependent degrees-of-freedom to independent degrees-of-freedom.
- PARAM,SHL2MEM: The new parameter SHL2MEM controls conversion of thin shell bending elements to pure membrane elements. All bending shells with thickness equal or less than the value of this parameter are automatically converted to membrane elements by the removal of the MID2 and MID3 data from the corresponding PSHELL data.
- PARAM,BUSHSTIF: The new parameter BUSHSTIF consists of two arguments which control the upper bound of translational and rotational stiffness for CBUSH elements. The corresponding stiffness values greater than BUSHSTIF(T/R) are reduced to specific values set by the parameter.
- PARAM,ELASSTIF: The new parameter ELASSTIF consists of two arguments which control the upper bound of translational and rotational stiffness for CELAS elements. The corresponding stiffness values greater than ELASSTIF(T/R) are reduced to specific values set by the parameter.

- PARAM,BUSHLMT: This parameter is used to issue a WARNING when the stiffness value for rotational components on the PBUSH entry exceeds the specified limit (BUSHLMT).
- PARAM,BUSHTLMT: This parameter is used to issue a WARNING when the stiffness value for translational components on the PBUSH entry exceeds the specified limit (BUSHTLMT).
- PARAM,ELASRLMT: This parameter is used to issue a WARNING when the stiffness value for rotational components on the CELAS2/4 or PELAS entry exceeds the specified limit (ELASRLMT).
- PARAM,ELASTLMT: This parameter is used to issue a WARNING when the stiffness value for translational components on the CELAS2/4 or PELAS entry exceeds the specified limit (ELASTLMT).
- PARAM,RECOVER: The new parameter RECOVER can be used to output the full-structure modes, within a specified frequency range, during Component Mode Synthesis (CMS) instead of the modes of the condensed system.
- PARAM,OP2GM34,TRUE/FALSE: The new parameter OP2GM34 can be used to control the output of the GEOM3 and GEOM4 data blocks to the .op2 file when PARAM,POST,-1 is specified.
- PARAM,AMLSMEM: The new parameter AMLSMEM can be used specify the amount of memory to be used by AMLS in Gigabytes. The default is the amount of memory used by OptiStruct.
- PARAM,FFRSMEM: The new parameter FFRSMEM can be used specify the amount of memory to be used by FastFRS in Gigabytes. The default is the amount of memory used by OptiStruct.

New Environment Variable

AMLS_MEM: This environment variable can be used to set the memory used by AMLS. It overrides the value set by the parameter PARAM, AMLSMEM.

Enhancements to the HyperWorks Solver Run Manager and HyperWorks Solver View Windows

A new Schedule delay option has been added to the HyperWorks Solver Run Manager window. It allows users to delay all jobs in the queue (which are not currently running) up to a predefined time (one week maximum).

A new Graph button has been added to the HyperWorks Solver View window for the Optimization Summary section. The Graph option allows the user to open a simple X-Y plot and browse the dynamic convergence of various variables displayed in the summary.

Enhanced Parameter Data

- PARAM,PRGPST: The parameter P RTPST can now be used to control the amount of AUTOSPC information printed to the .out file.
- PARAM, EXPERTNL: The AUTO option can now be used to activate a new "light" version of the expert system, which is designed to facilitate convergence of a nonlinear process in a reasonably close to minimum number of iterations.

Enhanced Bulk Data

- SOLVTYP: The ordering method for the MUMPS solver can be specified.
- Fully Qualified References (Parts and Instances): The following bulk data entries have been enhanced with support for fully qualified references. Selected local entries can now be referenced on these bulk data entries in a <PartName.number> format. Please see the corresponding *OptiStruct Reference Guide* and *OptiStruct User's Guide* sections for detailed information.
 - SPC, MPC, RELOC, MOMENT, FORCE, SPCD, SPCADD, MPCADD, RBE3, RBE2, RBE1, RROD, RBAR, CBUSH, and CBUSH1D.

- STRAIN: The MODAL option has now be added to request modal strain results for frequency response and transient analyses.
- BEGIN: The NAME= FEMODEL entity type has been added to indicate the beginning of a part. This is used in hierarchical modeling (parts and instances). The part has an arbitrary name (a maximum of 16 characters) and each part can be split in to multiple BEGIN/END blocks.
- END: The NAME= FEMODEL entity type has been added to indicate the end of a part. This is used in hierarchical modeling (parts and instances). The part has an arbitrary name (a maximum of 16 characters) and each part can be split in to multiple BEGIN/END blocks.

New Bulk Data

- INSTNCE: The INSTNCE bulk data entry can be used to define the location of a part in the global structure. This entry can reference a RELOC entry to map grid points from one location to another.
- RELOC: The RELOC bulk data entry can be used to move items or map grid points from one location to another. This entry allows the user to translate, match, rotate, or mirror grid points. The grid ID fields can be input as either numeric or fully qualified references to grid points in the model.
- CONNECT: The CONNECT bulk data entry can be used to define equivalence for all degrees of freedom of two grid points of two different parts within a specified tolerance. The tolerance is defined as the maximum distance between two grid points within which equivalence is allowed.
- PERBC: The PERBC bulk data entry can be used to define a connection between opposite edges/faces of the structure. A RELOC bulk data entry can be referenced on the PERBC entry to define the one-to-one match between the grids on both sides of the model.

New Subcase Data

- SUBMODEL: The SUBMODEL entry can be used in the Subcase Information section to select a submodel as a set of elements (and optionally a set of rigids). Subcase entries specific to the selected element set can be used to solve the submodel without affecting the rest of the structure.
- GLOBSUB: The GLOBSUB entry can be used in the Subcase Information section to select a subcase that references the global structure for Local-global Analysis. A set of grid points in the local structure that defines the transfer zone can also be specified.

Topology Optimization

DTPL Data MAXDIM Automatically Set if the Input Value is Too Low

If the input value of MAXDIM is too small, it will automatically be reset and an INFORMATION message issued.

Free Shape Optimization

Draft Angle can be Used With the Draw Constraint for Free Shape Optimization

In free shape optimization the nodes are free to move, so a smooth surface can be achieved for the draft angle.

Optimization of Multibody Dynamic Systems

Responses can be Generated for Particular Time Steps in Multibody Dynamics ESLM Optimization

Time step selection control for Multibody Dynamic Analysis based optimization is possible using ESLTIME and ESLTADD data. They can be used to specify time and response value ranges to select time steps for use in optimization. Additionally, the maximum or minimum response values and response peaks or troughs can also be specified for corresponding time step selection.

New Bulk Data

- ESLTIME: Used to specify the time and response value ranges of the responses
- ESLTADD: Used to combine multiple ESLTIME data

New Solution Control Data

- ESLTIME: Used to specify the set of ESLTIME and ESLTADD bulk data to be used to determine the time and response value ranges of the responses

OSSMOOTH

OSSMOOTH in HyperMesh can Generate 2nd Order Elements

The resulting mesh from OSSMOOTH in HyperMesh can contain 2nd order elements.

Internal Responses (DRESP1)

SPCFORCE can Now be Used as an Optimization Response

The DRESP1 data now supports the specification of SPCFORCE as a response using RTYPE = SPCFORCE.

GPFORCE can Now be Used as an Optimization Response

The DRESP1 data now supports the specification of GPFORCE as a response using RTYPE = GPFORCE.

External Responses (DRESP3)

DRESP3 External Responses can be Defined Using Microsoft Excel Files

The GROUP field on the DRESP3 entry can be used to specify the external Microsoft Excel workbook (via a LOADLIB entry) to define an external response. The CELLIN, C1#, CELLOUT, and CO fields on the DRESP3 entry can be used to specify the corresponding response values on specific Excel workbook cells. See the *External Responses User's Guide* section for detailed information.

Optimization Output

Design Sensitivity Analysis for Frequency Response in H3D

The sensitivities for displacement, velocity, and acceleration from Frequency Response Analysis are now output to the .h3d file. These sensitivities can be plotted vs. frequency in HyperGraph and are used by the Design Sensitivity Analysis utility.

Design Variable File for the Control of CMS Super Element Modal Information

For CMS super elements generated using the General Method, a design variable file can be generated with the DMIGDV continuation data of the CMSMETH bulk data. This ASCII design variable file can be included in an optimization run to design the modal (stiffness, mass, and damping) characteristics of the super element.

Enhanced Bulk Data Entries

CMSMETH: The DMIGDV continuation has been added to control the generation of an ASCII design variable file that can be included in an optimization run to design the modal (stiffness, mass, and damping) characteristics of the super element.

Optimization General Enhancements

Reliability-Based Design Optimization (Beta)

Reliability-based Design Optimization (RBDO) is a new beta feature that can be used to provide optimum designs in the presence of uncertainty. A gradient-based local approximation of responses is used to perform the Reliability Analysis. The design variables, constraints, and objective are tested for reliability based on user-defined reliability requirements.

Deterministic and random design variables can be selected for a design optimization run, while a random parameter is also available to check for robustness of a particular design within specified bounds. Similarly, design constraints and objectives can also be specified as deterministic (mean) or percentile values.

RAND and RANDP continuation lines on the DESVAR bulk data entry can be used to input design variables and design parameters, respectively. The ITYPE field can be used to select from a variety of random distribution types for Reliability Analysis. The design objective can be specified on the DESOBJ subcase information entry. The MAXP and MINP options can be used to specify percentile based calculation of the objective. The PROB option can be used to define the robustness requirement. Design constraints can also be set robustness requirement values using the PROB field on DCONSTR entries.

The OptiStruct implementation of RBDO uses local approximation for reliability assessment and is very efficient, and for most problems it yields reasonably accurate solutions. Local approximation based Reliability Analysis does, however, contain some error of varying degree. Accurate Reliability Analysis should be carried out if accurate satisfaction of reliability requirements is critical.

RBDO currently does not support topology, free-size and topography optimization.

Multi-Model Optimization

OptiStruct can now run multiple optimization models with common design variables in a single run. Multi-Model Optimization (MMO) allows you greater flexibility to optimize common components across

structures. Existing optimization models with common design variables can be optimized together without modifications.

The models can be included in the subcase section using ASSIGN, MMO, name, path.../filename.fem. OptiStruct MPI executables are required to run MMO and this feature is activated using the command line argument “-mmo”. The DRESPM continuation line on DRESP2 and DRESP3 bulk data entries can be used to reference responses from individual models. These responses can then be used similar to VARTYPE#’s on DRESP2 and DRESP3 entries to generate new responses.

DRESP2 can Reference DRESP1 with Different Numbers of Responses

The DRESP1L data can be used to pick a single response out of multiple responses to be combined with a single response.

Mode Tracking

- The printing of the mode tracking matrix is now controlled by PARAM,TRAKMTX.
- PARAM, TRAKMETH is now added to control the method used for mode tracking. The three methods that are available are the Mass Cross-Orthogonality Check (CORC), Modal Assurance Criterion (MAC), and Modal Assurance Criterion Square Root (MACSR).
- PARAM, MFLITER is now available for mode tracking. This parameter defines a threshold for the mode tracking matrix to check eigenvector correspondence. If an entry of the mode tracking matrix is greater than the specified threshold, the eigenvectors of previous and current iterations (corresponding to the row number and column number of the entry) are assumed to have correspondence.

ASSIGN,UPDATE Added CONM2 Data

CONM2 data that is changed during sizing optimization is now read in and updated when ASSIGN,UPDATE is used.

Default for VSUCC Changed to Zero in DSHUFFLE Data

The default value of VSUCC is now zero instead of one. VSUCC is now a percentage instead of a ratio.

Optimization with RADIOSS BLOCK Input Data

The entire structural optimization problem can be set up in the RADIOSS BLOCK input data format and then submitted to OptiStruct using the -rad command line argument. Please see the RADIOSS Release notes for a list of supported RADIOSS optimization entries and the *RADIOSS Optimization User’s Guide* section for detailed information.

New Parameter Data

- PARAM,TRAKMTX: The new parameter TRAKMTX controls the printing of the mode tracking matrix to the .out file.
- PARAM,TRAKMETH: The new parameter TRAKMETH is used to specify the Mode Tracking Method.
- PARAM,MFILTER: The new parameter MFILTER is used to specify the threshold for the mode tracking eigenvector correspondence.

Enhanced Bulk Data

- DESVAR: The RAND and RANP continuation lines have been added to specify distribution parameters for reliability-based design optimization. The ITYPE field can be used to select from a variety of distribution types.
- DCONSTR: The PROB field is now available to define the reliability requirement by specifying a probability value
- DTPL, DSIZE, and DRESP1: The Dang Van factor of safety for multi-axial fatigue has been added
- DSHUFFLE: The default value of VSUCC is now zero instead of one. VSUCC is now a percentage instead of a ratio
- DSHAPE: The DRAFT field has now been added to specify the draft angle (in degrees) for Draw Constraints (DRAW continuation line) in Free Shape Optimization
- DRESP3:
 - The GROUP field has been enhanced to allow the specification of an external Microsoft Excel workbook (via a LOADLIB entry) to define an external response. The CELLIN, CI#, CELLOUT, and CO fields on the DRESP3 entry can be used to specify the corresponding response values on specific Excel Workbook cells.
 - The DRESPM continuation line has been added to allow responses from multiple models to be referenced for (Multi-model Optimization) MMO.
- LOADLIB: The LOADLIB entry has been enhanced to reference a Microsoft Excel file for external response specification via DRESP3
- DRESP3: The DRESPM continuation line has been added to allow responses from multiple models to be referenced for Multi Model Optimization (MMO).

Enhanced I/O Options and Subcase Information Entries

- DESOBJ: The minimize percentile response (MINP) and maximize percentile response (MAXP) options have been added to the DESOBJ bulk data entry to specify the objective type. Additionally, the PROB argument can be used to define the reliability requirement by specifying a probability value.
- ASSIGN: The MMO Type is now available to include multiple models in a single optimization run. The user-defined model name and a path to the solver deck can be specified.

Nonlinear Dynamic Analysis

Nonlinear Dynamic Analysis in OptiStruct is suitable for highly nonlinear transient events (EXPDYN). In OptiStruct, it is implemented through an integration of the RADIOSS Starter and RADIOSS Engine using a translator. OptiStruct input data is directly translated into RADIOSS input data. The implementation is then carried out in RADIOSS and the results are transferred back into the OptiStruct output module. This functionality also partially supports Implicit Dynamic Analysis (IMPDYN) and Implicit Quasi-static Analysis (NLGEOM). Additionally, optimization support is available through the Equivalent Static Load (ESL) approach.

New Options Added for Extended CONTACT Property Type 7

The PCNTX7 data has been enhanced with FPENMAX (maximum fraction of initial penetration), DTMIN (limiting nodal time step), IREMGAP (flag to specify activation/deactivation of slave nodes), SENSID (sensor identifier) and four new options for the THEDAT heat contact data (FRAD, DRAD, FHEATS, and FHEATM).

New Extended Contact Property Type 24 (PCNTX24)

The PCNTX24 bulk data entry is a new extended contact property type 24.

Solid Element Properties Expanded to Support Navier Stokes Viscosity and an Advanced Tangent Matrix Estimation

The PSOLIDX data has been enhanced with the items LAMBDV, MUV, and IHKT to support Navier Stokes viscosity and advanced tangent matrix estimation.

Composite Element Material MATX25 Enhanced to Support Failure Based on the Ratio of the Number of Failed Layers to the Total Number of Layers

The MATX25 data has been enhanced with the item RATIO, which can be used to specify the ratio of the number of failed layers to the total number of layers that signifies failure of the element.

Rock-Concrete Material MATX21 Enhanced to Support External Pressure

The MATX21 data has been enhanced with the item PEXT, which can be used to specify the external pressure on the material.

Reaction Force Output at GRID is Now Available

The XHIST data now includes the REACTX, REACTY, REACTZ, REACTXX, REACTYY, and REACTZZ GRID output options.

Force-Deflection Curve Output from Snap-Through Analysis

For NLGEOM analysis, the Force-Deflection Curve from Snap-Through simulation can be generated using PARAM,SNAPTHRU,YES.

Weld Elements are Now Supported

CWELD and PWELD bulk data entries are now supported.

RADPRM, HMNAME can be Used to Preserve the Names Specified in the Bulk Data

RADPRM, HMNAME, YES can be used to preserve the component, property and material names defined by HM comments in the bulk data file and print them in the RADIOSS starter file after B2B conversion. This allows the retention of user defined names for entities after B2B conversion.

RADPRM, RBE2RBD can be Used to Control the Conversion of RBE2 Elements

RBE2 elements in OptiStruct are automatically converted into /RBE2 rigid elements. RADPRM, RBE2RBD, YES can be used to convert the RBE2 elements into /RBODY rigid bodies.

Responses can be Generated for Particular Time Steps in ESLM Optimization

Time step selection control for optimization is possible using ESLTIME and ESLTADD data. They can be used to specify time and response value ranges to select time steps for use in optimization. Additionally, the maximum or minimum response values and response peaks or troughs can also be specified for corresponding time step selection.

DRESP2 can Reference DRESP1L and DRESP2L for Optimization

The DRESP2 data can be used for optimization by referencing the corresponding nonlinear subcases via DRESP1L and DRESP2L.

MU1 Value on CONTACT/PCONT can Now be Used for FRIC on Corresponding /INTER Entries

If FRIC is not explicitly defined on the PCONTX/PCNTX# entries, the MU1 value on CONTACT/PCONT is now used for FRIC in the /INTER entries. Otherwise, FRIC on PCONTX/PCNTX# overwrites MU1 values on CONTACT/PCONT.

New Parameter Data

- PARAM,SNAPTHRU: PARAM,SNAPTHRU,YES is used to control the generation of a Force-Deflection curve from Snap Through analysis. The default is NO.
- RADPRM,HMNAME: The new parameter HMNAME controls the conversion of the component, property and material names.
- RADPRM,RBE2RBD: The new parameter RBE2RBD controls the conversion of RBE2 elements.

Enhanced Bulk Data

- PCNTX7: The PCNTX7 data has been enhanced with FPENMAX (maximum fraction of initial penetration), DTMIN (limiting nodal time step), IREMGAP (flag to specify activation/deactivation of slave nodes), SENSID (sensor identifier) and four new options for the THEDAT heat contact data (FRAD, DRAD, FHEATS, and FHEATM).
- PSOLIDX: The PSOLIDX data has been enhanced with the items LAMBDAY, MUV, and IHKT to support Navier Stokes viscosity and advanced tangent matrix estimation.
- MATX25: The MATX25 data has been enhanced with the item RATIO, which can be used to specify the ratio of the number of failed layers to the total number of layers that signifies failure of the element.
- MATX21: The MATX21 data has been enhanced with the item PEXT, which can be used to specify the external pressure on the material.
- XHIST: The REACTX, REACTY, REACTZ, REACTXX, REACTYY, and REACTZZ GRID options can now be used to output reaction forces.

New Bulk Data Entries

- PCNTX24: The PCNTX24 Bulk Data Entry is a new extended contact property type 24 added.
- ESLTIME: Used to specify the time and response value ranges of the responses.
- ESLTADD: Used to combine multiple ESLTIME data.

New Solution Control Data

- **ESLTIME:** Used to specify the set of ESLTIME and ESLTADD bulk data to be used to determine the time and response value ranges of the responses.

Enhanced I/O Options and Subcase Information Entries

- **DESOBJ:** The minimize percentile response (MINP) and maximize percentile response (MAXP) options have been added to the DESOBJ bulk data entry to specify the objective type. Additionally, the PROB argument can be used to define the reliability requirement by specifying a probability value.
- **ASSIGN:** The MMO Type is now available to include multiple models in a single optimization run. The user-defined model name and a path to the solver deck can be specified.

Resolved Issues

- The accuracy of the PENTA element has been improved through a new element formulation.
- In certain situations, discrete design optimization was not activated when TMANUF is specified on the PLY entry. This issue has been fixed and discrete design optimization is now activated upon specification of TMANUF.
- A possible internal programming error during Modal Transient Analysis runs with static loads, using the Lanczos solver, has now been fixed.
- A possible programming error during flexbody generation (Craig-Chang formulation) when SPC's are used has now been fixed.
- When a Transient Thermal Analysis was run with very small time steps, negative temperatures at some grid points were observed. This issue has now been resolved.
- The compliance of SUBCOM subcases that linearly combine static subcases (with SPCD/non-zero SPC loading) was incorrect. This issue has now been resolved by triggering element-wise compliance calculations.
- The Component Mode Synthesis (CMS) superelement analysis can now be run if the input file doesn't include any grids
- An improved error message is now output when IDENT is specified on the ACMODL entry and the fluid-structure interface grid points are controlled by shape design variables with non-zero initial perturbations.
- OptiStruct has now been enhanced with long integer support, allowing for the use of memory sizes above 16 GB. This resolves an issue where memory overflow error occurs if AMPFFACT is set to a high value.
- An issue where OptiStruct automatically resets the user-defined negative value of GE on the CELAS entry to -1 has now been fixed.
- ESLM-MBD topology optimization runs weren't converging due to incorrect ESL calculations when penalties change over iterations. This issue has now been resolved.
- Von Mises ply stresses and strains are now output to the H3D format when CSTRESS/CSTRAIN output is requested to H3D.
- During ESL (NLGEOM) optimization runs, the incorrect time step was selected. This issue has now been resolved.
- In a B2B run, ply stresses/strains are now converted from elemental coordinate system to ply coordinate system and output to the H3D file. These ply stresses/strains in the ply coordinate system can be used to calculate the composite ply failure indices.

- The function of INTOL on the ACMODL entry has been improved to fix incorrect peak amplitudes at certain frequencies in an acoustic cavity.
- The SURF entry definition has been updated to resolve an issue when nodes on the section surface for PRETENS definition (3D pretension) are the same as the nodes used to define the contact between the bolt surface and the wall of the hole.
- MODALSE output is now available regardless of whether any standard output is requested.
- Residual vector output was not output if a DLOAD bulk data entry is referenced by a DLOAD subcase information entry using AMSES or AMLS. This issue has now been resolved.
- A possible internal programming error during Frequency Response Analysis runs with panels defined using ERPPNL and PFPANEL output request, has now been fixed.
- Resolved an issue where the entire model was exported to the SIMPACK (.fbi) file regardless of the specifications on the MODEL card.
- The PROPERTY label is now supported on field 3 of the DSA entry.
- An issue where OptiStruct incorrectly output an error when the M2GG was specified within a subcase has now been resolved.
- Previously, the solver GUI listed the last used solver options and input file regardless of the selected solver type. This issue has now been fixed and the solver GUI will start with last used input file/options for the selected solver.
- An error was incorrectly output for stress/strain responses on elements/properties associated with linear materials in NLSTAT loadcases. This issue has now been resolved by improved error checking to verify that such elements/properties are associated with a nonlinear material.
- A possible error in the SUBSEQ entry while reading a single positive number at the start of a continuation line has now been resolved.
- The number of rigid body modes output from an AMSES run was lower than expected. This issue has now been resolved.
- An issue where the tensor of complex stress results from a Frequency Response Analysis was incorrectly visualized using the H3D file has been resolved by enforcing default systems in the solver.
- An error was incorrectly output when elements different from shells were input using the ELIST entry. This issue has now been resolved by removing the error and allowing non-shell type elements on the ELIST entry and these elements are skipped from the analysis.
- An issue where SPCD unloading effects are not properly accounted for in Nonlinear Dynamic Analysis has now been resolved.
- A programming error occurred, for some models, when AMSES was run out-of-core. This issue has now been resolved.
- Incorrect sensitivities were output during free-size optimization runs for ERP responses (on skinned solids), where the design space was modeled with TRIA6 elements. This issue has now been resolved.
- Incorrect sensitivities were output during topology optimization runs for ERP responses, where the design space was modeled with PENTA15 elements. This issue has now been resolved.
- USET,U6 RESVEC are now calculated when Lanczos is used for Normal Modes Analysis.
- MFLUID results could be wrong if both sides of the panel were wetted surfaces. This has been resolved.
- Unreferenced MFLUID data could lead to ERROR 2894. This issue has been resolved.
- Negative SPC ID were displayed in the .out file. This issue has been resolved.
- Now zero nodal thickness for shell elements is allowed.
- The handling of mechanisms in the model by AMSES was improved in order to capture all the rigid body modes
- Flex bodies cannot be generated when there is directional mass in PUNCH DMIG used in the flex body model. The data PARAM,CMSDIRM,YES must be specified to allow the use of PUNCH DMIG with directional mass.

- LIMIT1 data is not required on the MATS1 entry if TABLES1 or TABLEST data is specified.
- Warning messages are now printed if mesh distortion occurs as a result of using ADJUST on the CONTACT and TIE bulk data entries (except if ADJUST is used for S2S contact).
- Displacement results are now excluded from the .mnf file during flexible body generation (PARAM, ADAMSMNF) if DISPLACEMENT(H3D)=NONE is specified.
- The in-plane bending behavior of the CQUAD4 element has now been improved.
- RROD, RBAR, RBE1, RBE2, and RBE3 are now available in the .mnf file during flexible body generation (PARAM, ADAMSMNF).
- AMSES runs with very high frequency upper bound values (V2 field on EIGRA) and low number of roots (ND field on EIGRA) values can now be solved.
- The results from PARAM, COMP2SHL, YES will now be output to the screen if SCREEN, OUT is specified.
- An error message is now output if super element content ID's change from positive to negative as a result of negative ID shifts specified on DMIGMOD data.
- An error message is now output if USETYPE=CB/CN is specified on the CMSMETH entry during component mode synthesis.
- The composite ply orientation angles are initialized through /INISHE/ORTH_LOC instead of /INISHE/ORTHO for NLGEOM, IMPDYN, and EXPDYN analysis.
- The composite ply orientation angles are initialized through /INISHE/ORTH_LOC instead of /INISHE/ORTHO for NLGEOM, IMPDYN, and EXPDYN analysis
- The HyperWorks Solver View (Run Manager) retains the last used size settings of the window.

RADIOSS 13.0 Release Notes

[RADIOSS for Crash and Safety](#)

13.0 release includes major enhancements for airbags deployment. New contact with specific treatment for penetrations and intersection which allows nodes on the wrong side to be released during the airbag inflation. A new 4 nodes membrane element with full integration improves the aspect of the inflated airbag surface resolving the out-of-plane hourglass; it has been designed for fabric materials, LAW19 and LAW58. Enhancements in the advanced non-linear anisotropic fabric material LAW58 make it easier to identify input parameters for a perfect fit with tests data. Porosity of the fabric can also be taken into account, both for external surfaces and internal stripes or structures.

TYPE7 auto-impacting contact definition extended to mesh size smaller than gap value; enhanced stability of TYPE11 with incremental friction formulation; tied contact with asymmetric tension/compression failure; coupled tension shear failure model for spotwelds; enhanced robustness of XFEM method for cracks propagation modeling; advanced open cells foam material law taking into account air flow thru the cells.

[RADIOSS for Metal Forming](#)

Advanced ultra high strength steel material law for hot-forming (LAW80); NXT failure criterion; friction energy converted into heat for both TYPE7 and TYPE21 contact; drawbeads (contact TYPE8) with bending force input improves prediction of force on blank; damage initialization for solids.

[RADIOSS for Blast Simulation](#)

Multi-material LAW51 compatibility with Drucker-Prager yield criterion allows modeling underground explosions; new equation of state P-alpha to model materials porosity; several new features have been implemented, like detonation lines, wave shapers, Johnson-Cook Sesame material law with possible conductive and radiative transport in plasma.

[RADIOSS - General](#)

New contact TYPE24 allows solid to solid contact definition with zero gap; AMS enhanced performance thanks to compatibility between /DT/AMS and /DT/NODA/CST (applying AMS on a given part or set of parts and regular mass scaling on the remaining parts); enhanced scalability with HMPP version of Multi-Domain; Optimization based on Equivalent Static Lad Method using OptiStruct; User Subroutines compatibility with dynamic libraries; new tetra4 element formulation (ltet=3) compatible with ALE.

Introduction

Altair® RADIOSS® is a leading structural analysis solver for highly non-linear problems under dynamic loadings. It is highly differentiated for scalability, quality and robustness, and consists of features for multiphysics simulation and advanced materials such as composites.

RADIOSS is used across all industry worldwide to improve the crashworthiness, safety, and manufacturability of structural designs. For over 20 years, RADIOSS has established itself as a leader and an industry standard for automotive crash and impact analysis.

RADIOSS for Crash and Safety

Airbags

- **TYPE23 contact:** Penalty formulation with constant stiffness, specifically designed for airbags fabric self-contact. Initial penetrations and intersections are allowed and resolved automatically during the airbag inflation. It is a node to surface contact, edges are not treated.

- **Membrane element:** 4 nodes membrane element with full integration (based on Batoz formulation) compatible with fabric material laws; it prevents hourglass. For airbag modeling, recommendation is to couple it with total strain formulation, which reduces strain accumulation errors.
- **Reverse flow:** For special purposes it is sometimes necessary to compute inflow thru vent-holes. This is now available both for Uniform Pressure and FVM methods (Chemkin model).
- **Lost heat flow:** Heat transfer coefficient can be used to calculate lost heat flow due to convection.
- **Fabric properties:** Fabric porosity can be prescribed thru several leakage models (/LEAK/MAT); they are available both for UP and FVM methods. Porosity can be prescribed both for an internal surface or an external one (including vent holes).
- **Fabric material laws:** Advanced fabric material LAW58 has an extended formulation describing with higher accuracy the mechanical behavior of fabric at high strain values; new tabulated input enables account temperature dependency to be accounted for.
- **Reference state based on elements:** /EREF: Elements based reference state; new input type, based on element IDs.
- **Resolved issues:**
 - REFSTA and XREF can now be used within a single model
 - MONVOL/AIRBAG1, PROP/INJECT1 & 2, MAT/GAS compatibility with //SUBMODEL
 - MONVOL/FVMBAG1 + DT/NODA/CST: could lead to divergence due to negative volume
 - FVM: general robustness improvements (better control on the evolution of the number of finite volumes)

Contacts – TYPE24

- Node-to-element contact based on penalty formulation with constant stiffness; this formulation allows large time-steps, as opposed to contact TYPE7, which may severely affect time-steps.
- Formulation is essentially more robust and more tolerant (than TYPE7) to model imperfections, like initial penetrations and intersections.
- TYPE24 manages solid to solid contact with a zero gap (different from TYPE7, in which a gap must be defined).
- Performance and scalability in parallel mode are optimized and comparable to TYPE7.
- Compatible with AMS.
- Initial penetrations treatment, with possible release of nodes with initial penetrations.
- Available for Non Linear Implicit (currently not compatible with MPI parallel mode).
- Resolved issues:
 - Poor performance with single precision
 - Node impacting a solid element edge: wrong master segment was found
 - Large sliding: while crossing edges, moving from one solid element to neighbor contact could be lost
 - TETRA10 contact stiffness adjustments (improved nodal forces balance between corner and middle nodes)

Contacts – TYPE7

- Self-contact with gap larger than mesh size – self-contact with shell modeling could not be treated when mesh size is smaller than gap value. With the new algorithm, neighboring nodes to a master segment are automatically removed from the list of contact candidates (Irem gap = 2).
- Geometrical definition of master surfaces (lcurve): now available for both concave and convex master surfaces.
- Friction energy can now be converted into heat.

- Resolved issues:
 - TYPE7 + Istf=1: prescribed stiffness was not taken into account
 - In case of solids coated with shells elements, contact stiffness could be based on solids instead of shells
-

Contacts – TYPE11

- New incremental friction formulation: improves contact stability and allows larger time-steps at the same time.
-

Contacts – TYPE2 (TIED)

- Search distance for master segments identification: new option (Ignore=3).
-

Contacts TYPE3, 5, 7, 10, 11, 19, 20

- New flags (Idel=-1 or -2) for nodes and segments deletion allow to keep non-connected slave nodes.
-

Multi-Domain

- **Scalability:** SPMD parallelization of RAD2RAD has been achieved and displays tremendous speed-up for a high number of cores compared to previous SMP RAD2RAD. It can also run in Hybrid SMP/SPMD mode.
 - **Kinematic conditions:** compatibility has been extended to cylindrical joints.
 - **Output:** single TH file can be output. Sections across domains are now available for output.
 - **Known limitations:** /MPC ; /RBE2 ; /GJOINT ; /ALE (unless inside a domain) ; Idel and Idel2 flag not passing at domain border ; /DAMP ; /AMS ; /SENSOR types: gauges, inter, rwall: not yet synchronized and in any case: /SPH ; FX_BODY ; /MODIF.
 - Resolved issues:
 - Fixed contact stiffness issue with orthotropic materials
 - Sections across domains were ignored
-

XFEM

- Brand new, more powerful and a more efficient XFEM formulation. It is compatible with 4-node and 3-node shells, both:
 - Single layer (PID1)
 - Multi-layer (PID11) – temporarily up to 3 layers
- Fully compatible with hybrid parallel mode; the following failure criteria are XFEM enabled:
 - /FAIL/FLD
 - /FAIL/TAB1
 - /FAIL/JOHNSON
 - /FAIL/TBUTCHER
- Resolved issues:
 - Single layer and multilayer compatibility in a single model
 - Results consistency between SMP, SPMD and HMPP parallel modes
 - Crack visualization issue in case XFEM + PID11 + RWALL + SPMD

Note: In order to improve usability, XFEM input is now a more compact format and plugged directly into the failure criteria listed above; initial crack definition was also changed for improved ergonomics: /LEVSET has been replaced with /INICRACK.

Spotwelds

- /MAT/LAW59: physical thickness parameter has been introduced; geometrical description of the spotweld improves accuracy in shear behavior.
 - /FAIL/CONNECT: possibility to deactivate failure in compression.
 - /MAT/LAW83: advanced connection material law; yield stress can be defined as a user function coupling normal and tangential direction.
 - /FAIL/SNCONNECT: coupled tension/shear failure model, compatible with LAW59 and LAW83.
-

Tabulated Failure Criteria (/FAIL/TAB)

- /FAIL/TAB: now compatible with solids; failure curves can be prescribed as a function of Lode angle.
 - /FAIL/TAB1: this is the same as /FAIL/TAB but with TABLE input instead of functions.
-

Composites

- /ANIM/BRICK/ORTHD: output of orthotropic directions for thick shells and solids.
 - QEPH compatibility with composite shells.
 - Resolved issues:
 - Field 10 (E33) is ignored for /MAT/LAW25
 - LAW25 + ANIM/SHELL/FAIL + lshell=12: output of failed layer was wrong
 - LAW25 + PID9 + stress tensors output were null
 - Stress and strain TH output for orthotropic solid and thick shell elements
-

Pulley Spring

- Variable friction input can be used to lock safety belt pretensioners.
-

Foams

- **LAW70:** Compatibility with Isolid=17 (Iframe=1 & 2); resolves hourglass deformation.
 - Compatibility with total strain formulation (Ismstr=11)
 - **LAW77:** This is an advanced material law for foams, taking into account the behavior of the air inside the foam cells. The air flow thru the cells affects the foam mechanical properties, like stiffness, viscosity, and so on. Special boundary conditions are provided to bound the cells porosity in case of contact with non-porous parts.
-

Polymers

- **LAW76 (SAMP):** Strain rate dependency and table input.

Metals

- **LAW36:** Any number of functions is now allowed with new format (previously limited to 20 functions).

Resolved Issues

- If several XELEM are sharing the same property results are wrong.
- XELEM output issue in SPMD parallel mode.
- MAT/LAW65: material does not follow prescribed curve in loading and in unloading.
- LAW59: wrong calculation in plastic region for low E values.
- BEAM TYPE18: time step calculation for circular section with IP=1.
- /PROP/SPR_AXI rupture inactive for Ifail2=2.
- FAIL/TAB + tetras: Elements do not fail.
- FAIL/TENSSTRAIN failure model not compatible with 2D quad elements.

RADIOSS for Metal Forming

Hot-Forming - Material LAW80

This law allows the modeling of the ultra-high strength steel behavior at high temperatures and the phase transformation phenomena from austenite to ferrite, pearlite, bainite, and martensite during cooling.

CONTACT TYPE21

- Improved performance.
- Friction energy can now be converted into heat.
- Non-uniform nodal temperature can be prescribed.
- Thickness defined at /PART level is taken into account.

CONTACT TYPE8

- Bending force input improves prediction of force on blank.
- Improved starter performance.

NXT Failure Criterion

- Similar to FLD, but based on stress instead of strain.

Thermal Time-Step

- Computed to ensure stability of thermo-mechanical simulations.

Damage Initialization (with .sta file)

- /STATE/BRI/FAIL: export variables for damage initialization.
- /INIBRI/FAIL: initialize the damage.

Resolved Issues

- LAW78: resolved potential instability issue.
- LAW73: engine stops when extrapolation from tables output a yield value less than 0; now RADIOSS always considers a yield value greater than or equal to 0.
- STATE output for TETRA10 in .sta file.

RADIOSS for Blast Simulation

P-alpha - MATERIAL LAW75

- Porous material law to take into account the effect of porosity on the equation of state.

Tetra4 Improvements

- S-FEM technique implemented to improve results quality with tetra4 in ALE.

MATERIAL LAW51

- Compatibility with Drucker-Prager yield criteria for soil modeling.
- Post-processing plasticity and temperature with standard keywords:
 - /ANIM/ELEM/EPSP for plasticity
 - /ANIM/ELEM/TEMP for temperature
- Improved silent boundaries formulation for blast applications.
- Strain rate and temperature issues fixed in the Johnson-Cook criterion.

Post-Processing

- Material Tracking for ALE multi-material laws (LAW20, LAW37 & LAW51)
 - /ANIM/ELEM/VFRAC - allows tracking of material evolution; it is targeted to replace global density contour, which was not always adequate
 - /ANIM/ELEM/BFRAC – enables output of burn fraction for explosive materials
- Nodal output
 - /ANIM/NODA/P for pressure
 - /ANIM/NODA/DENS for density

Grid Velocity Formulation Parameters

ALE parameters standardization; the following keywords are available both in Starter and Engine:

- /ALE/DONEA
- /ALE/DISP
- /ALE/SPRING
- /ALE/ZERO
- /ALE/STANDARD

Grid Velocity Links

- It is now possible to define links on grid velocity in Starter input:
 - /ALE/LINK/VEL
- ALE links can be activated or deactivated by using /ALE/LINK/ON and /OFF.
- It is now possible to use groups of nodes to define links in the Engine file using the new Engine keyword /ALE/LINK/VEL. The previous keyword /VEL/ALE is now obsolete.

Upwind Method (in the Engine)

- Similar to /UPWIND in the Starter, it enables to change the upwind parameters during a restart.
- /UPWM/SUPG for streamline upwind Petrov-Galerkin for momentum advection; it is now compatible with LAW51; fixes potential issues with velocity results.

Defense Options

- /DFS/DETLINE enables the definition of a detonation line.
- /DFS/WAV_SHAP wave shapers enabled to take into account obstacles during the detonation wave evolution.
- /MAT/SESAM Johnson-Cook Sesame material law with possible conductive and radiative transport in plasma. The Sesame tables are not provided by Altair.
- /DFS/LASER enables the modeling of laser impact taking into account laser-matter interaction.

Resolved Issues

- Format checks for LAW3, LAW4, and LAW49: Equation of State is no longer included in material cards; must be referring to /EOS.
- Drucker-Prager yield criteria fixed in tension.
- /TH/BRIC/EPSXX does not work with law4.

Optimization

RADIOSS Optimization provides RADIOSS users an easy way to set up optimization problems, take advantage of optimization solutions inside OptiStruct, and simultaneously use RADIOSS for analysis. The equivalent static load method (ESLM) is used to perform the optimization run.

An extra input file (<name>.rad) apart from the usual Starter and Engine input files is required to run RADIOSS optimization. Optimization entities, like optimization objectives, constraints, design variables, responses, and so on, are defined within this file. The `-rad` run option is used to run the <name>.rad input file and OptiStruct then looks for the RADIOSS Started and Engine files within the same working directory. Updated Starter and Engine files are written out by OptiStruct for each optimization iteration and they are solved by RADIOSS. The corresponding output is then communicated back to OptiStruct to update the optimization model. This iterative process is repeated until convergence and the optimized model is output.

This feature allows users to combine the unique analysis capabilities of RADIOSS with the optimization features of OptiStruct to generate a optimized designs. Topology (/DTPL), Topography (/DTPG), Sizing (/DESVAR), Free-Shape (/DSHAPE), and Free-Sizing (/DSIZE) optimization types are supported.

AMS

- Kinematic conditions compatibility extended to moving rigid walls.
- Compatibility between /DT/AMS and /DT/NODA/CST allowing the application of AMS only to a group of parts and a classical time step control to the remaining parts of a model for optimized global performances.
- Improved robustness (new convergence criterion).
- Performance improvement for /DT/INTER/AMS.
- Known limitations:
 - 2D analysis
 - /RBE2
 - /RBE3
 - Lagrange Multiplier
 - /LAW13
 - /RIVET
 - /MPC
 - /GJOINT
 - /FXBODY
 - SHEL16
 - /BRICK20
 - /NSTRAND
 - RBODY activated by sensors
- Resolved issues:
 - /DT/AMS : cannot keep the target time with /MAT/VOID + tied interface
 - AMS & /DAMP + Kinematic conditions: kinematic conditions might not be fully ensured
 - AMS and Penta6 elements caused Engine failure

Single Precision – Solid Elements

- Improved accuracy of single precision with solids and QEPH shell elements.

SENSORS Improvements

- Activation/deactivation of contacts.
- Possibility to stop a job; allows tool positioning.
- Resolved issues:
 - Imposed displacement was not respected if controlled by sensor
 - KJOINT2: sensor is not working in /PROP/KJOINT2

Starter Warnings and Error Messages

- Title of the option is printed in the .out file.
- Starter errors out in the following cases:
 - an empty group is detected
 - a vent hole surface is not defined

Engine Error Messages

- In case of mass or energy error:
 - The list of nodes with highest velocity is output. Scope is an easier identification of the nodes (parts) responsible for the divergence.
 - An ANIM file is automatically written.

Elements

- Integrated Beams (TYPE18): it is possible to refer to predefined sections (rectangular or circular).
- Tetras: new flag lmas allows redistributing nodal mass according to element shape (corner angles), which improves robustness.
- TETRA4: new lrot flag activates S-FEM formulation, which is recommended for ALE.
- Degenerated hexa elements time step optimization: default changed into ON (/DTSDE).
- Total strain formulation reduces errors accumulation is available for:
 - /PROP/TYPE9 (SH_ORTH) Orthotropic shells
 - /PROP/TYPE14 (SOLID) for:
 - Isolid=14 (HA8): full integration, variable number of IP
 - Isolid=17 (H8C): full integration, 2 x 2 x 2 IP
 - Isolid=24 (HEPH): under-integrated, with physical hourglass stabilization

User Subroutines

- Compatible with dynamic libraries.
- Compatible with orthotropic materials.

SPH

- Pressure gauge for SPH.
- SPH compatibility with //SUBMODEL.
- Compatibility with Johnson Holmquist material law.
- Reduced memory usage at the Starter level by using Metis Multilevel Kway domain decomposition; it allows to run in SPMD parallel mode very large SPH models (> 10M particles).
- Resolved issues:
 - Domain decomposition fails if NUMSPH*KVOISPH > 2exp31 for large SPH models
 - Possible core dump with SPH symmetry conditions in SPMD parallel mode
 - Compatibility with Modif files

Geo-Material

- **/MAT/LAW81**: Drucker-Prager with strain hardening cap model and dilatancy; compatible with ALE.

Post-Processing

- **/ANIM/GPS/TENS**: Outputs the six components of stress on node locations, by extrapolation from integration points.

Transformation Matrix

- Compatible with transformations available in CAD systems and/or multibody models.

Damping Fine Tuning

- **/DAMP/INTER**: Sensitivity analysis.

/SURF/GRBRIC/EXT and /SURF/GRBRIC/FREE

- Extracts the external (or free) surface of a group of solid elements.

Starter Performance and Memory Optimization

- Starter parallelization and memory consumption optimization.

//SUBMODEL

- SUBMODEL format defined in the header (of the submodel) cards has the priority on the (optional) format defined at the //SUBMODEL card level; it improves ergonomics in case submodels with different formats are to be included in a single model.

- Resolved issues:
 - Encrypted cards compatibility
 - SPH compatibility
 - MONVOL/AIRBAG1, PROP/INJECT1 & 2, MAT/GAS compatibility
 - MAT/FOAM_TAB and PROP/SPR_PUL: incorrect functions identifiers offsets are applied
-

Implicit

- LAW12 and LAW14 compatibility with implicit solution.
 - Compatibility with moving frames.
 - Improved performance in SMP with new version of MUMPS solver.
 - Non Linear: Improved convergence of MAT/LAW27.
 - Resolved issues
 - In case of gravity setting, nodal velocities were not reset to zero after implicit run; the following explicit run could have been affected
-

Resolved Issues

- Running in SPMD a first job with /DT/NODA/CST and then switching to AMS caused engine failure.
- Incorrect orthotropy results using /INIBRI/ORTHO.
- /INISHE/ORTH_LOC : values in degrees were not available.
- /INIVEL/AXIS and /INIVEL/TRA incompatibility.
- RBE3 with many independent node crashes.
- RBE3 brings added mass from independent nodes to dependent ones.
- Issue in visualization of RBE2 and RBE3 if SMPD > 1.
- /NLIMPL+ moving skew caused an error for the implicit solution.
- /ANIM/TENS/STRESS/ALL caused engine error.
- /ANIM/SHELL/FAIL: element shown as failed while it is not.
- /ANIM/SHELL/TENS/STRESS/UPPER and ../2 and ../3 output were null for orthotropic materials (LAW25 + PID9).
- Incorrect stress tensor output for orthotropic PIDs (PID 9, 10, 11 and 16) and material laws ≥ 28 (i.e. LAW28, LAW29, ...).
- Wait time extended for RADIOSS Starter to prevent error when using Templex to resolve /PARAMETER cards.
- Multiple /INIVOL cards definition conflicts.
- /INVOL not compatible with Modif file.
- Timestep calculation when using LAW34.
- /LOAD/PFLUID calculation of force is smaller than theoretical value.
- /PROP/SPR_AXI rupture inactive for Ifail2=2.

MotionSolve 13.0 Release Notes

Introduction

MotionSolve is a state-of-the-art multibody solver available in HyperWorks. It has a complete set of modeling elements and powerful numerical methods to support a full set of analysis methods. The accuracy, speed and robustness of MotionSolve have been validated through extensive testing with customer models and test data. MotionSolve also offers unmatched compatibility with ADAMS/Solver input. [Click here for more information about MotionSolve.](#)

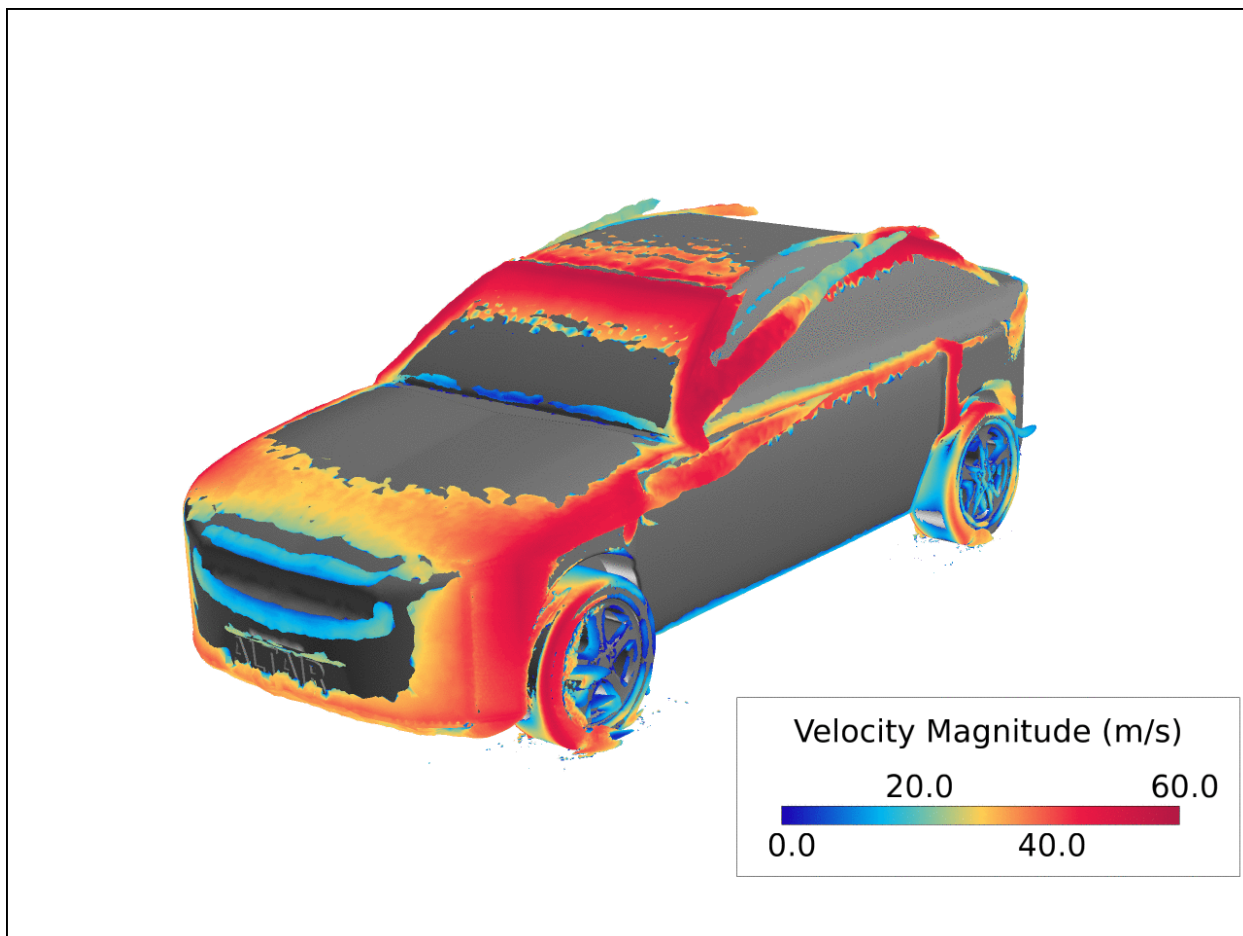
This document describes changes that have occurred to MotionSolve since version 12.0. All of these improvements are available in the 13.0 version of HyperWorks.

Core Solver Enhancements

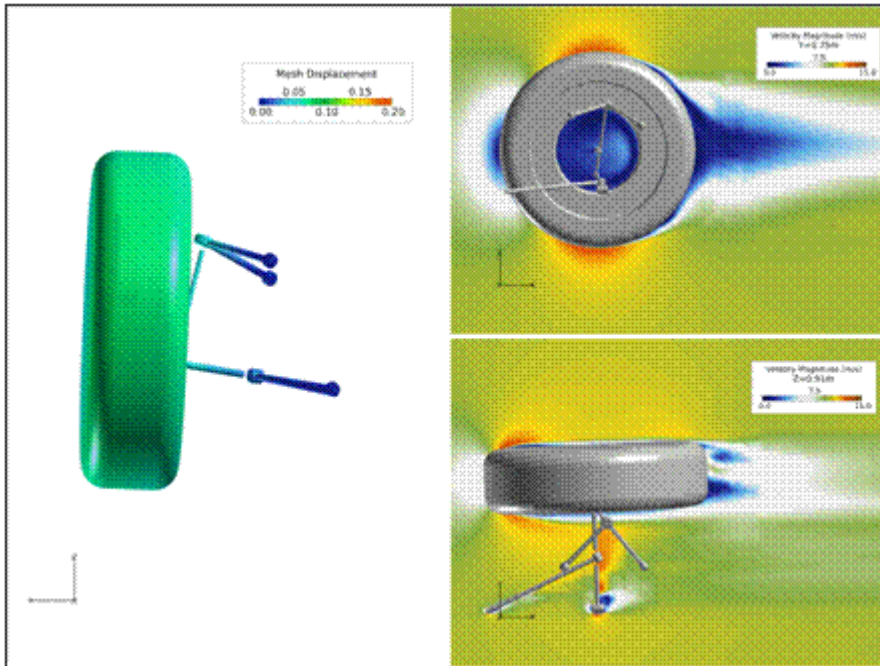
Directly Coupled MBD-CFD Simulations: Co-simulation with AcuSolve

With this release, MotionSolve now supports co-simulation with AcuSolve, Altair's CFD solver, which allows you to couple your multibody system model with a corresponding fluid flow model. This coupling allows you to simulate the effects of fluid loads on the rigid bodies in your multibody system and vice-versa. MotionSolve and AcuSolve are coupled via a socket connection, allowing both solvers to run on different systems.

Shown as an example here are the velocity contours for flow over a full vehicle and a quarter car suspension. The fluid flow induces a force on the exterior surfaces of the vehicle which in turn affects the suspension height of the car. The change in height, in turn affects the fluid force on the vehicle.



Iso view of velocity contours for flow over a 4 wheel vehicle



Air flow over a quarter car suspension

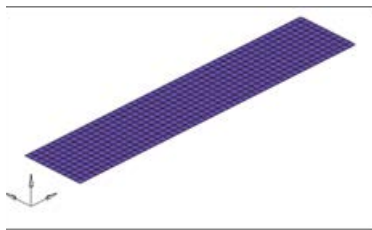
Enhanced Gravity Modeling

The Force_Gravity modeling element can now be specified as a function expression as well as a real number. This allows you to model gravity as a function of time in your model.

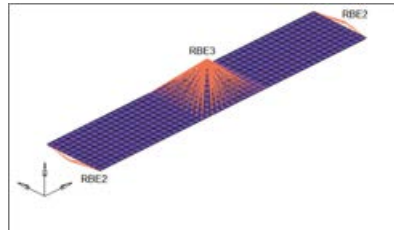
Enhancements to Animation .h3d Files

This release also contains several enhancements to the animation .h3d file written out by MotionSolve:

- RIGID elements present in the CMS flexbody file are now written into the animation H3D for better visualization

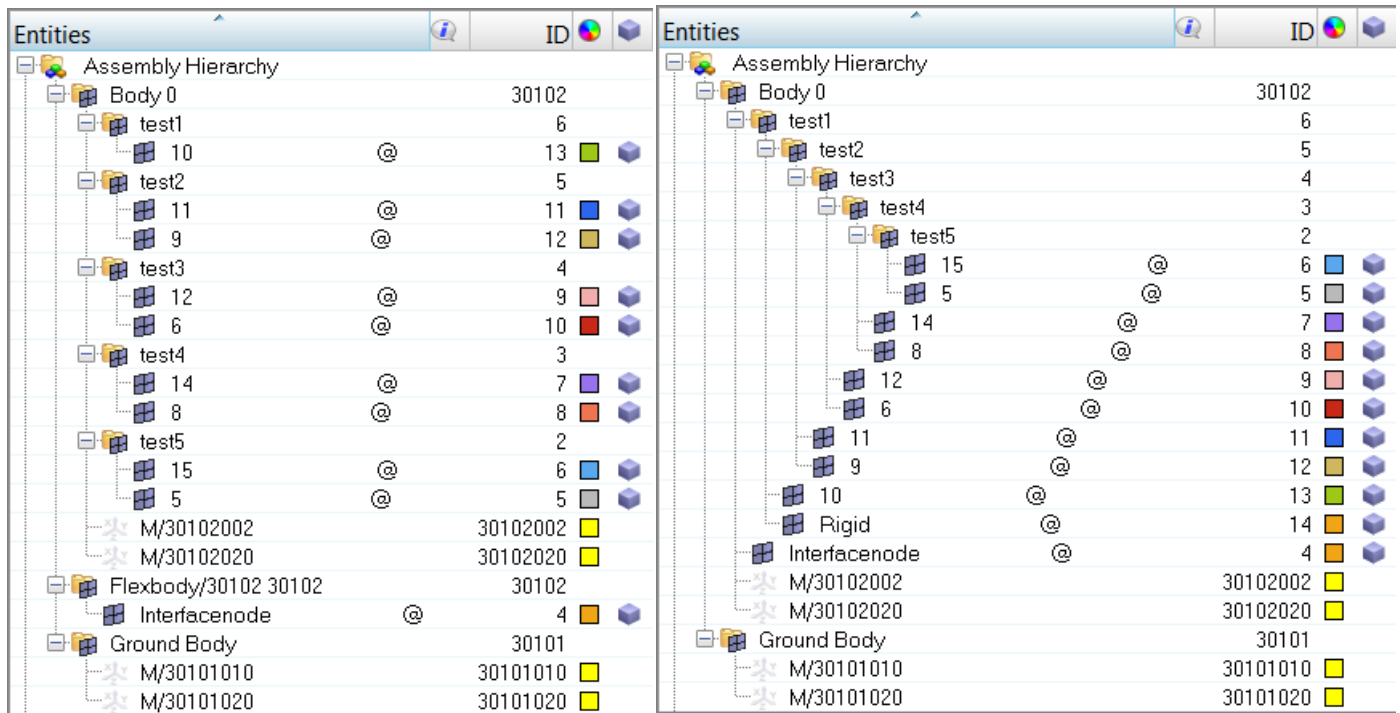


H3D generated by MotionSolve 12.0



H3D generated by MotionSolve 13.0

- Shell thickness information in the flex .h3d file is transferred to the animation .h3d file for better visualization
- Assembly hierarchy, if present while creating a CMS flexbody, is transferred to the animation .h3d file for consistent visualization between the pre- and post-processor



MotionSolve 12.0

MotionSolve 13.0

Enhancements to Assembly Analysis

The criteria for automatically triggering an Assembly Analysis between simulations has been updated for cases when the model has flexible bodies. The new criteria detects changes to the model configuration (including flex bodies) and accordingly triggers an Assembly Analysis when required.

This change calculates realistic body accelerations when there are multiple simulates in models that contain flex bodies.

Enhancements to Point to Deformable Surface (PTdSF) Model Element

Three key enhancements have been made to the current implementation of the PTdSF modeling element.

- The first change improves the robustness of models containing PTdSF elements. It was observed that in the corrector iterations, the surface parameters U and/or V could go out of the range defined for the surface and the simulation would stop. In the enhanced implementation, if the U or V parameters begin to go out of range, MotionSolve prints out a warning and continues with the simulation. The U or V parameters are held fixed at the start or end limits until they come back into range. If the model is well defined, the U or V parameters will come back into range.
- The second change increases the generality of PTdSF. PTdSFSUB – the user subroutine definition for the PTdSF model element has been enhanced to accept SYSFNC and SYSARY calls. This means that the force in a PTdSF element may now be a function of other model states. This is particularly useful for defining complex force models inside the PTdSFSUB. For instance, the coefficient of friction may be calculated using a differential equation. This state dependent value may be subsequently accessed in the PTdSFSUB.
- The third change improves the performance of the PTdSF element. The relevant DAE constraint scaling terms in the Jacobian have been optimally scaled to improve performance, with improvements up to 70% in the solver analysis time.

Enhancements to Solver Robustness

A new scheme for choosing the time step size during the end of the simulation has been implemented such that the situation where the integrator selects a very small step size to reach the end time is avoided. This avoids spurious spikes in the results due to small step sizes chosen by the integrator.

Faster Simulation Times While Using SI 1

The solver has been enhanced to provide faster simulation times when using the DSTIFF integrator with DAE index 1 (Stabilized Index 1). The scaling of the Jacobian terms corresponding to constraint equations has been modified. This has resulted in faster simulation times for SI1. Speed improvements between 10-15% are seen on average.

Improved Solver Messages

The solver messaging has been improved in some areas. This enables you to better understand what the solver is doing and it allows you to more easily debug your model. Some of these include:

- Improved error message if non-matching parenthesis are detected in any function expression
 - Improved, easy to understand warning message when redundant constraints are detected in the model
 - Improved, easy to understand warning messages when a U or V parameter goes out of range in a PTCV, PTdCV, PTSF, PTdSF etc. modeling element
 - Improved, easy to understand error messages when a negative stiffness or damping value is specified for a spring damper or bushing modeling element
 - Easy to understand error message when a joint or joints associated with a Constraint Coupler model element are not found in the input deck
 - User written messages: More options are available for the user while issuing user written warning/error/info/fault messages. For more details, please refer to the documentation on USRMES
 - More accurate error messages are printed out when an evaluation of a MotionSolve expression fails
 - When an action in a Sensor Event element is triggered, MotionSolve writes out output information to the screen only once as long as the sensor is active – this helps in making the output/log file easy to read. This behavior can be controlled via the verbose attribute in the Sensor_Event element.
-

Improved Workflow for Co-simulation Using Simulink Coder™

The workflow for co-simulation between MotionSolve and Simulink® via the Simulink Coder™ has been revised for ease of use. In the current version, you can make use of a script to compile your Simulink Coder™ model into a dynamic linked library.

The new workflow is illustrated below:



This script compiles and links the code generated by Simulink Coder™ automatically, making it ready to use with MotionSolve for co-simulation. Please refer to the MV-7005 tutorial for more information.

Kinetic, Strain and Dissipative Energy Distribution for Linear Analysis

MotionSolve now writes out the modal kinetic, strain and dissipative energy distribution for Linear Analysis. This information is available both in the solver log file as well as in the *_linz.mrf output file that is generated at the end of the analysis. You can set the *write_energy_dist* attribute to TRUE in Param_Linear to enable this feature.

This feature can benefit users in DOE/Optimization studies. For example, within the MotionView – HyperStudy framework, you can now perform optimization studies on powertrain mounts accounting for modal purity and mode spacing.

Modeling Check for Spring Damper, Bushing, Beam and Field Elements

Previously, if you specified I and J markers belonging to the same body for the above model elements, MotionSolve would fail during the analysis with little information about the cause. This made debugging the model difficult. With this release, an additional check has been added. If the I and J markers for the above elements belong to the same body, a relevant error message is printed out and the simulation is stopped.

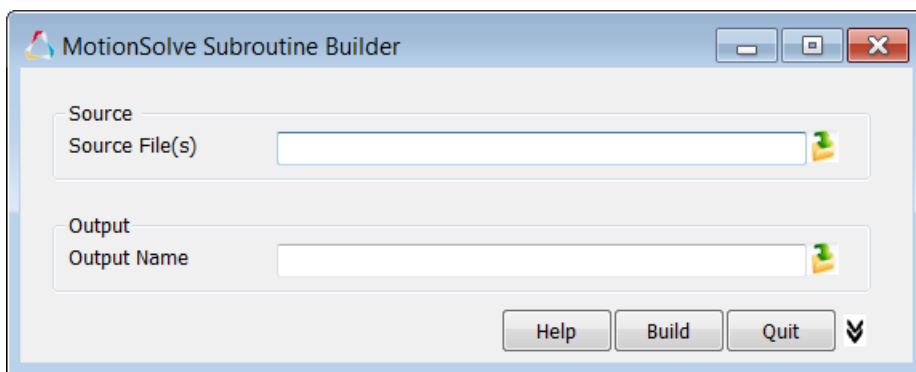
Python User Subroutine Library

In this release, MotionSolve provides a library of user subroutines written in Python for your reference. These can be accessed at <altair_root>\hwsolvers\motionsolve\usersub\py_src.

Additionally, the environment variable MS_USERDLL_DIR may now be used to locate your Python scripts. This allows you to place your Python user subroutine scripts in a different location from your MBD models without having to specify the full path of the script in the XML input file.

Redesigned Windows User Subroutine Build Tool

This release contains a completely re-designed user subroutine build tool for Windows and a similar new tool for Linux.



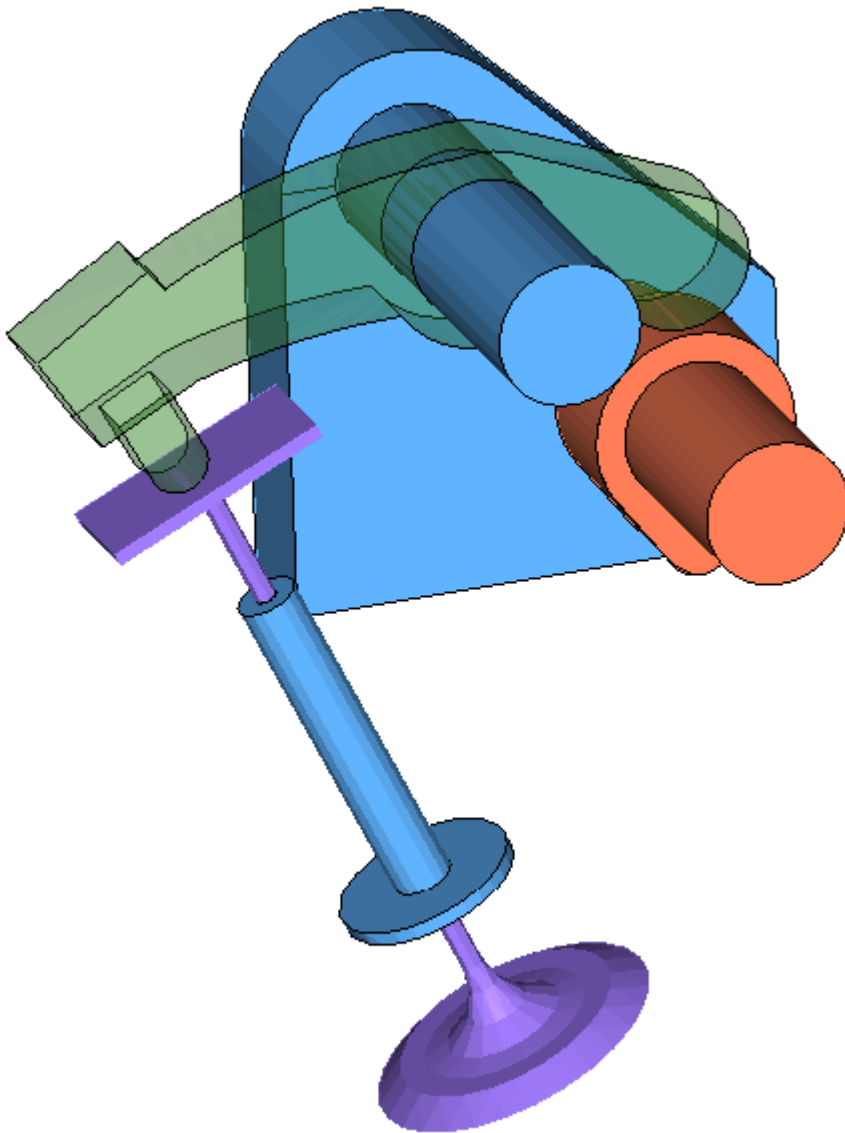
The user-subroutine build tool will compile and link your user written source code so it can be used in a MotionSolve simulation. Some key features of this tool are:

- Support for compiling and linking C/C++/Fortran source code and object files
- Automatic configuration of compile/link options based on compiler availability on user's machine
- Auto generation of Microsoft Visual Studio solution files (if available on user machine)
- Lightweight, easy to use tool that requires minimum user clicks
- Consistent look and feel between Windows and Linux platforms

Visualizing Rigid Body Contact

With this release, the default setting for visualizing rigid body contact forces in HyperView has changed. Now, if rigid body contact is modeled, MotionSolve will always write out rigid body contact information allowing you to visualize the contact force vectors in HyperView. To turn this feature off, you may modify the `contact_gra_output` attribute within the `ResOutput` command.

The figure below illustrates contact vectors that are drawn in HyperView when contact is detected between two rigid bodies in a MotionSolve simulation. The contact vectors indicate the point of contact and the direction and magnitude of the contact force.



Contact force vectors generated using MotionSolve 13.0

Resolved Issues

Contact Force Visualization

While visualizing contact force vectors in HyperView, the force vectors are now correctly drawn to originate from the point(s) of contact instead of the body CG.

Python User Subroutines

Previously, you were unable to run a simulation from the MotionSolve GUI if your model contained a Python user-subroutine. This has been fixed within this release.

User Defined Graphics

Custom graphics defined in a GRASUB could not be visualized in HyperView. With this release, this issue has been fixed.

Deactivating Joints

Previously, if you deactivated a joint between simulations, any motions associated with that joint would still be active which could lead to erroneous results. Now, in such a situation, any motions associated with a deactivated joint are deactivated automatically and a warning message is printed out to alert the user of the same.

TIMGET Utility Function

The TIMGET function always returned "0" as the current time when it was called from a user-subroutine during a static or quasi-static simulation. This has been fixed.

Forced Assembly Analysis and Initial Velocity Calculation

Previously, the initial velocity calculations were reset if you forced an assembly analysis before a transient simulation. This led to incorrect initial velocities in the subsequent transient analysis. The correct behavior is to re-calculate initial velocities after a forced assembly analysis. This is implemented in the current release.

Motion of Type Velocity in Static Analyses Permitted

Previously, you were only able to prescribe a motion of type displacement within a Static Analysis. Specifying a motion of type velocity caused an error message and the simulation to halt. Now, if a motion of type velocity is specified, MotionSolve fixes the motion value and the simulation continues.

Using VARVAL to Access Co-simulation Variables

Results obtained from using VARVAL(..) while accessing Plant Input variables could be noisy. This has been fixed such that using VARVAL(..) or PINVAL(...,..) gives the same result.

Changing Force Units for Transient Analyses

MotionSolve produced inconsistent results when the force units were changed from Newton to Kilogram_Force. This was observed in models that contained CMS flexible bodies. This issue had been resolved within the current release.

End Time Mismatch Between MotionSolve and Simulink® Model in Co-simulation

When running a model in co-simulation with Simulink (using a library generated by Simulink Coder™), if the end times between the MotionSolve model and the Simulink model were not the same, the co-simulation would hang. This issue has been fixed and the new behavior is such that the end time for the Simulink model is adjusted automatically by MotionSolve to be the same as the end time for the MotionSolve model.

AutoSpring Setup Using Force as Install Method

Previously, if you chose force as the installation method for an AutoSpring and entered the spring force at the design position, you would not obtain that force during the simulation. This issue has been resolved so that at design position, the spring generates the force specified.

Loading Binary .crg Road Files with FTire in MotionSolve

Previously, you were unable to load a binary road file (*.crg) for your FTire tire model in MotionSolve. This has been addressed as part of the current release.

CD Tire Auxiliary Outputs

Previously, auxiliary outputs for the CDTire model (slip angle, slip ratio, and so on) were reported incorrectly in a MotionSolve simulation. This has been fixed such that the correct values are now reported in the output files.

Save/Load Command Causes Solver Failure

For the following cases, issuing a Save followed by a Load command caused the simulation to fail:

- When using the DSTIFF integrator with index 1
- When the model contains flexible bodies
- When the model contains PTdSF model elements

This has been fixed for all the cases above within this release.

SPLINE_READ User Subroutine

In the previous versions of MotionSolve, you were unable to specify a relative path for your Python script containing the SPLINE_READ user subroutine in MotionSolve. This has been fixed within the current release.

Simulating Joint Friction on Linux Platform

In version 12.0, you were unable to simulate joint friction in your model on any Linux machine. This was due to an issue with loading the required library correctly. This has been fixed within this release allowing you to model joint friction on both Windows and Linux platforms.

PLT File Generation for Models Containing User Graphics

In version 12.0, MotionSolve did not write the .plt file to disk if your model contained a user defined graphic. This has been fixed within this release.

Documentation

A number of errors in the documentation have been resolved for both model and command elements.

ADAMS Compatibility

Additional Support for Model Units

More options for choosing the model units are now available. Some of these include MegaNewton, Poundal, US Ton, Yard, Nanosecond, Day, and so on.

Fixes to Model Translation from adm/acf to xml

This release contains several fixes for translating model elements from the ADM/ACF input deck to MotionSolve's XML format:

- MFORCE elements with the SCALE attribute defined as a function expression are translated correctly.
- FRICTION statement is translated correctly.
- Multiple LINEAR commands were incorrectly translated. This has been fixed.
- The GSE statement with line breaks in the ROUTINE attribute was incorrectly translated. This has been fixed.
- In the comment part of a REQUEST statement, the character "@" was translated incorrectly. This has been fixed.
- The usage of the functions FX, FY, FZ, TX, TY and TZ in MotionSolve are now consistent with their respective definitions in ADAMS® C++ Solver.

Automotive Extensions to the Solver

MotionView and MotionSolve support an extensive set vehicle simulation tools.

- MotionView now includes new vehicle dynamics preference. New entities are available for modeling bushings, springs, dampers, rebound-stops, bump-stops, tires and drivers.
- MotionSolve contains new DLLs (or shared libraries) that capture the behavior of these entities.
- The tools support activities in the areas of:
 - Suspension kinematic and compliance (K&C) analysis

- Vehicle dynamics
- Durability
- Ride Comfort

These entities support Tiemorbit property files along with reuse of common Adams/Car™ property files with your MotionView and MotionSolve models.

The MotionView online help describes how to add these entities to your model. Use of the vehicle dynamics tools requires installation of HWSolvers 13.0.

In MotionSolve, the following are common to all modeling elements:

- An equal but opposite force is applied on two bodies that these entities are defined between.
- The force applied depends on the displacement and velocity of one body relative to the other.
- Outputs specific to each entity are supported, for example tire lateral slip angle for the tire entity.
- The parameters (for example stiffness), used to compute the force are stored in a property file that is independent of the MotionView MDL file and the MotionSolve input deck. The property files are:
 - Text files and may be viewed and edited using any common text editor
 - Compatible with Adams/Car – property files from Adams/Car may be reused with the Auto Entities in MotionView
 - Read by MotionSolve just prior to analysis

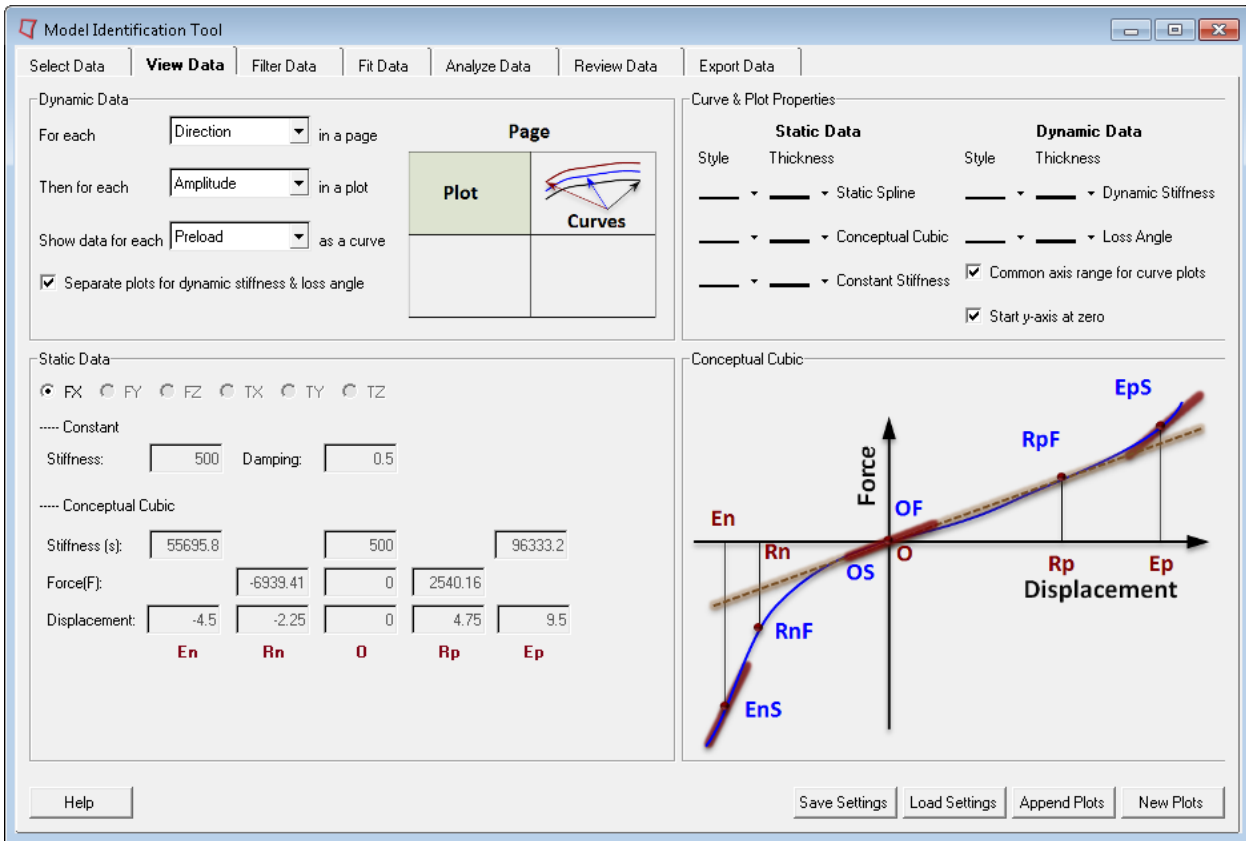
A complete description of enhancements, bug fixes and other changes to automotive extensions may be found [here](#).

The Altair Bushing Model

HyperWorks HW13.0-HWDesktop contains new features for supporting frequency, and amplitude dependent bushings. The implementation consists of four different components that work together. Collectively, they provide a comprehensive capability for identifying bushings from test data and, subsequently, using bushing models in a simulation. The components are summarized below.

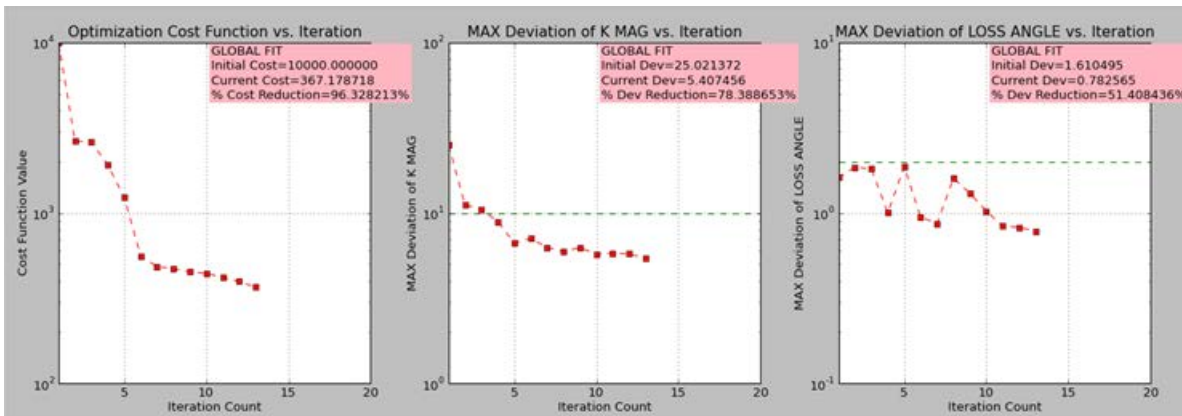
Model Identification Tool

The MIT manages the workflow associated with fitting experimentally-measured bushing data to analytical bushing models and storing these in ASCII property files. The models in this context are for frequency, amplitude and preload dependent bushings. Two dynamic and three static models are supported. The MIT is launched from HyperGraph.



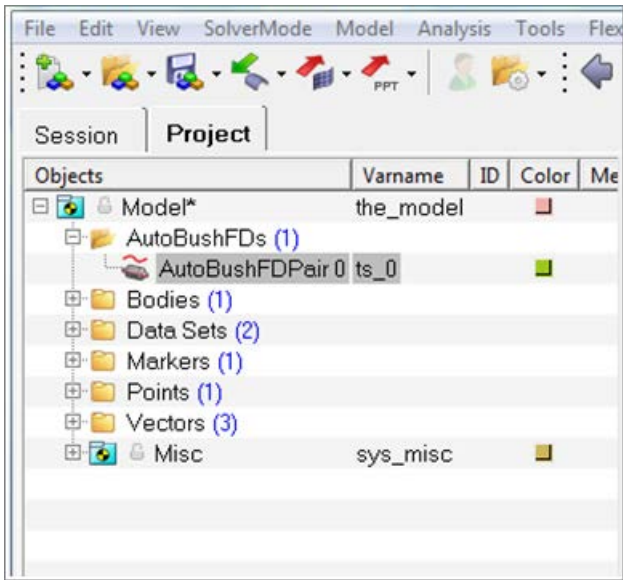
MIT Fitting Tool

This tool is invoked by the MIT user interface. It applies sophisticated numerical methods to calculate model coefficients so that deviations between model behavior and known experimental data are minimized. The fitting tool is an extension to HyperGraph.



Bushing Model in MotionView

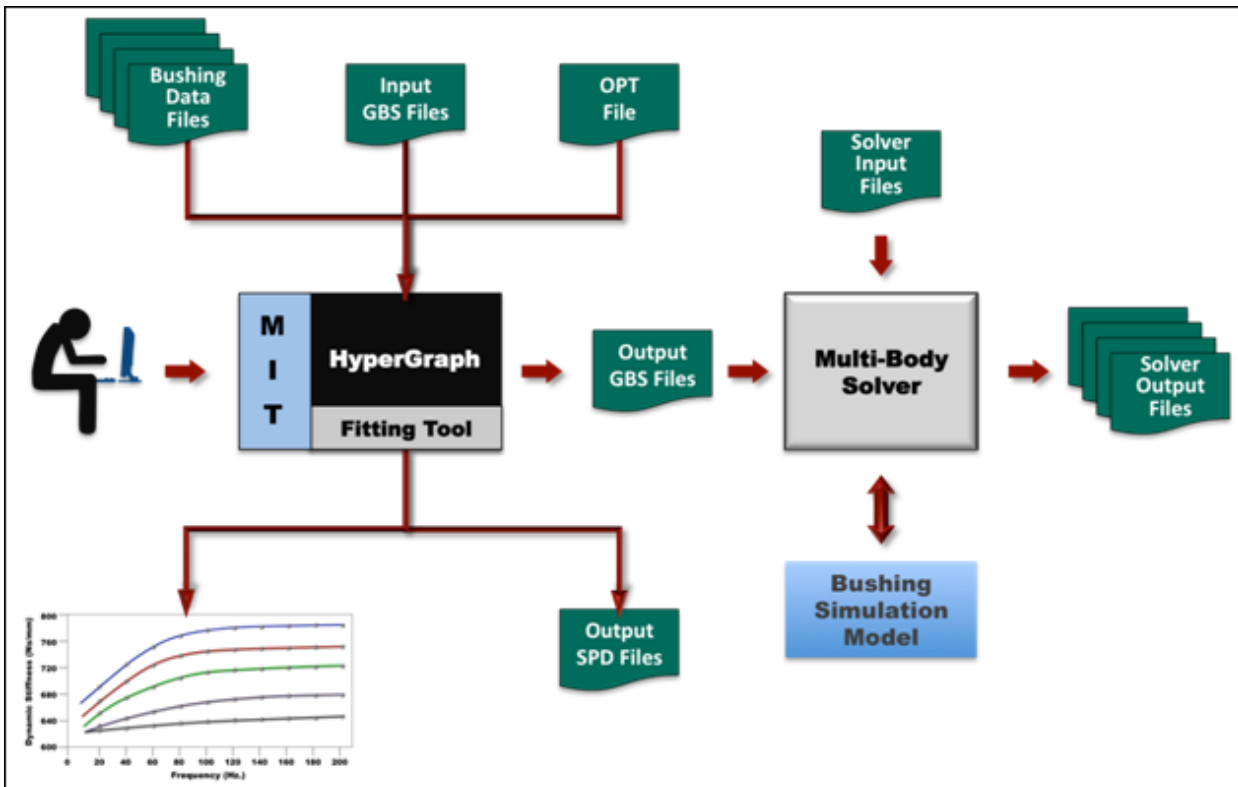
This is a set of panels in MotionView to instantiate and modify frequency, amplitude and preload dependent bushings in a vehicle model.



Altair Bushing Simulation Model

The Altair Bushing Simulation Model is used with fitted bushing models in a multi-body simulation. It is a solver run-time demand load library (.dll or shared library) that works with MotionSolve. Adaptors for Adams/Solver™, MATLAB™ and SIMPACK™ are also available.

The use of this new capability is illustrated in the figure below.



The user interaction with this system is as follows:

- The user interacts with the MIT to fit bushing test data to analytical models. The output of the fitting process is a GBS file that contains the coefficients for the analytical models. The GBS is thus an analytical representation of the test data.
- A full vehicle model is built in MotionView. Bushings are used as flexible connectors at various places in the vehicle.
- The model is passed to MotionSolve, the MBD solver. At run time, MotionSolve loads the Bushing Simulation Model dll (or shared library) to evaluate bushing behavior.

- The output from the solver is a set of files that describe the complete response of the vehicle for the test that was performed.

For more detailed information, please see the release notes for the [Model Identification Tool](#).

AcuSolve 13.0 Release Notes

The 13.0 release of the AcuSolve product suite continues to build on Altair's class leading solver technology. The latest release of the software delivers new technologies to expand capabilities and improve usability. The 13.0 release provides improvements to AcuSolve's moving mesh technology, new multiphysics coupling in the form of multi body dynamics, and a newly developed validation manual. In addition to these new features, a number of critical bugs have been addressed in this release, further improving the quality of the product for end users. The details of the release are provided in the following sections

Introduction

The V13.0 release of the AcuSolve product suite comes equipped with many new features, improvements, and bug fixes. This release continues to build on our class leading CFD solution by providing new technology for multiphysics and moving mesh applications, improvements to smooth and rough wall turbulence modeling, and a validation manual that documents the solver's accuracy for a variety of applications.

13.0 Release Notes

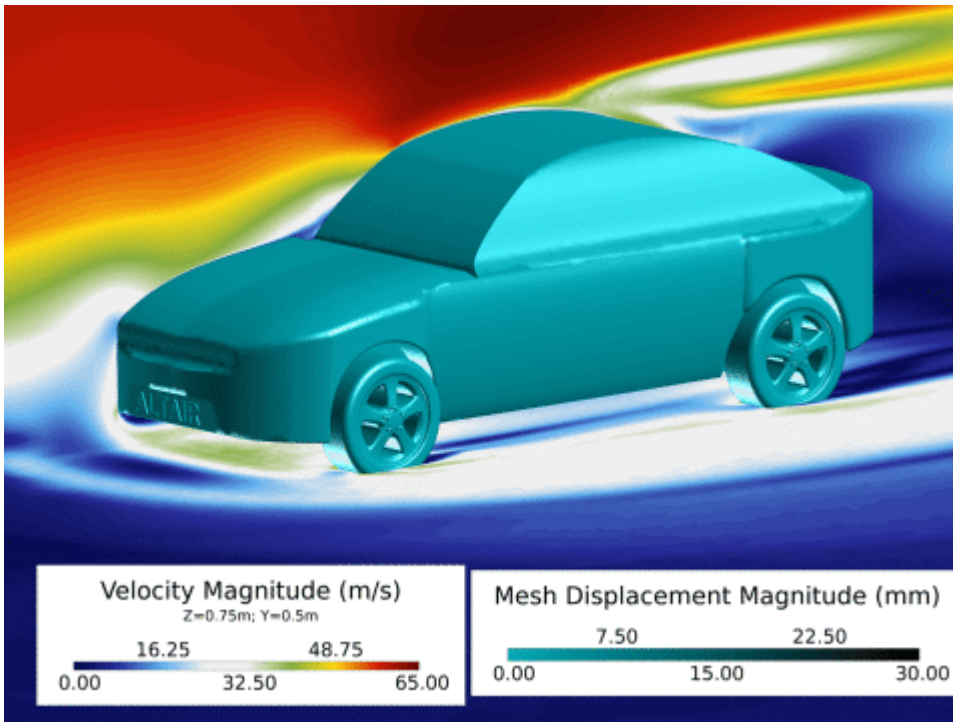
Directly Coupled Multibody Dynamics Simulations using AcuSolve/MotionSolve

AcuSolve now supports bi-directionally coupled multibody dynamics simulations using MotionSolve. This feature provides the ability to solve multiphysics simulations that involve rigid body motion of complex mechanisms by using MotionSolve to simulate the motion of the mechanical system and AcuSolve to provide loading on the components of the rigid bodies. This feature expands upon AcuSolve's existing rigid body dynamics solver by adding the following capabilities:

- The ability to simulate multibody dynamics models with linkages
- The ability to simulate single- (or multi-) body dynamics models with complex physics that are not accurately modeled by the internal rigid body solver
- The ability to simulate full mechanical systems

The `EXTERNAL_CODE` and `EXTERNAL_CODE_SURFACE` commands have been expanded to facilitate this coupling. AcuSolve and MotionSolve communicate through a socket connection, allowing the user to run each solver on a different system. Note that use of this feature requires special care in the definition of the AcuSolve model. Since MotionSolve has no geometric representation of the model, it is necessary for users to indicate which AcuSolve surfaces are assigned to each part in the MotionSolve model. Additionally, the physical representation of the geometry within the AcuSolve model must be such that the action of any idealized linkage does not cause interference between physical surfaces in the AcuSolve model. For example, a four-bar mechanism that contains revolute joints and pin connections must be modeled such that space is left between the parts to enable relative motion of the components in the fluid model without collision of the surfaces.

The applications of the AcuSolve/MotionSolve coupling range in complexity from simple mechanisms up to full vehicle aerodynamics/suspension interaction simulations. An example application showing the response of a vehicle's suspension to wind loading is shown in the following animation.



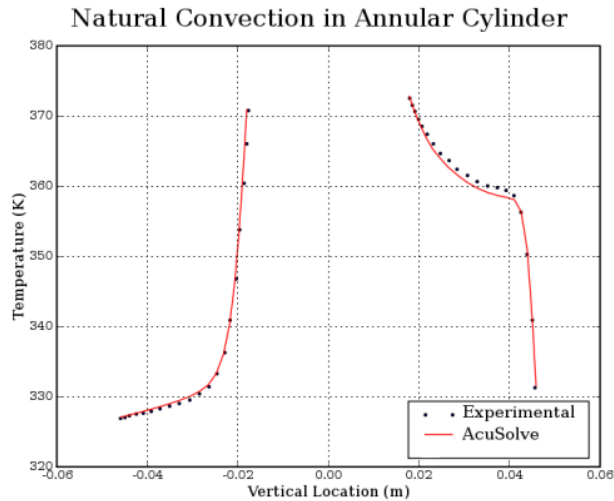
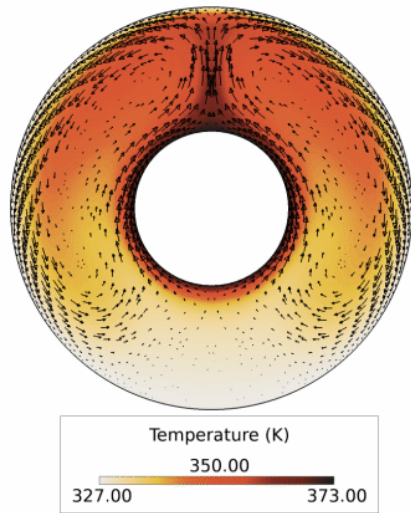
Animation showing the suspension movement in response to wind loading on a vehicle

The AcuSolve/MotionSolve interface supports the use of AcuSolve's proprietary Multi-Iterative Coupling (MIC) strategy and is thus capable of simulating tightly and loosely coupled interactions. Note that this feature, and technology in general, is in its first release. With this in mind, users are encouraged to provide feedback to Altair regarding their desired applications and experiences in using it.

Introduction of the AcuSolve Validation Manual

The AcuSolve installation now comes equipped with a validation manual that documents the accuracy of the solver for various applications. The purpose of the document is to provide fundamental validation of AcuSolve in comparison to analytical results or commonly referenced experimental data. The validation manual provides a brief overview of each validation case and a high level description of the relevant settings necessary to simulate the case with AcuSolve. The results of each simulation are presented and compared to the experimental or empirical results.

The manual is available from the AcuSolve Help Home page. All files necessary to run the simulations that are described in the manual are also available in the distribution within the following directory: `<installation_directory>/model_files/validation/<case_name>`. The current version of the manual includes 15 cases and will be expanded upon in future releases of the code.



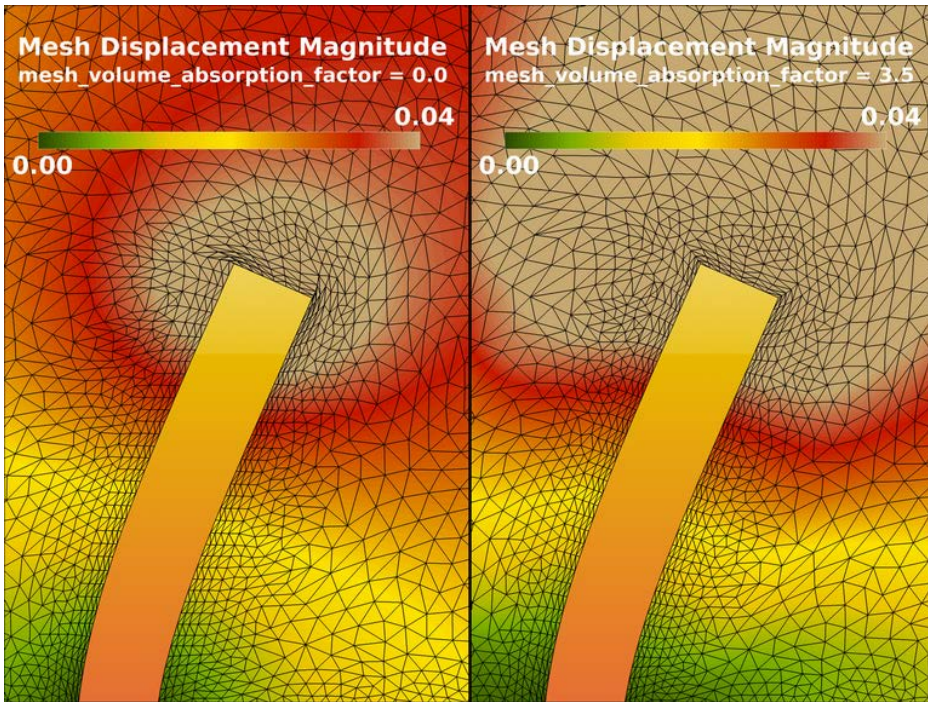
Representative results from validation manual showing AcuSolve solution compared to experimental data for natural convection in an annular cylinder.

Volume Based Element Stiffness in ALE Mesh Motion

AcuSolve's Arbitrary Lagrange-Eulerian (ALE) mesh motion technology has been enhanced to allow an alternate formulation for computing the stiffness of each element. In past releases, the stiffness of a given element was assigned based on the proximity to the motion. In general, this leads to behavior where the elements closest to the motion exhibit high levels of deformation regardless of their size. In many cases, this behavior is not desirable. To circumvent this behavior, the element stiffness can now be computed based upon a relative volume weighting among the elements in the model. Using this approach, the smaller elements are assigned higher stiffness values. Users have control over how heavily weighted the smaller elements are through a newly introduced command line option to AcuSolve. Note that this feature is currently deactivated by default (set equal to 0.0). To activate the feature, users should set the following cnf parameter to a value greater than 0.0:

```
mesh_volume_absorption_factor = 2.5
```

Higher values for the `mesh_volume_absorption_factor` yield greater differences in stiffness between the small and large elements in the model. In practice, this encourages the small elements to remain rigid while the large elements absorb the majority of the motion. Note that the optimal value for this factor is somewhat case dependent. However, internal testing has shown that values ranging from 2.0 - 4.0 work well for a broad range of problems. Use of this option does introduce a significant change in the matrix structure of the ALE equations, which can lead to increased CPU time. The increased compute time does come with improved robustness for many applications though. This option is particularly beneficial for applications that contain high levels of bending, as shown in the following figures. Note the improvement in the shape of the elements at the tip of the beam for the case that utilizes the `mesh_volume_absorption_factor=3.5`. The improved element quality supports larger deformations of the beam by propagating the mesh motion further from the body before it is diffused. This concept is illustrated by the contours of mesh displacement magnitude. The elements at the tip of the beam are near collapse in the left hand image (using default ALE settings) while the element shape is greatly improved in the right hand image.



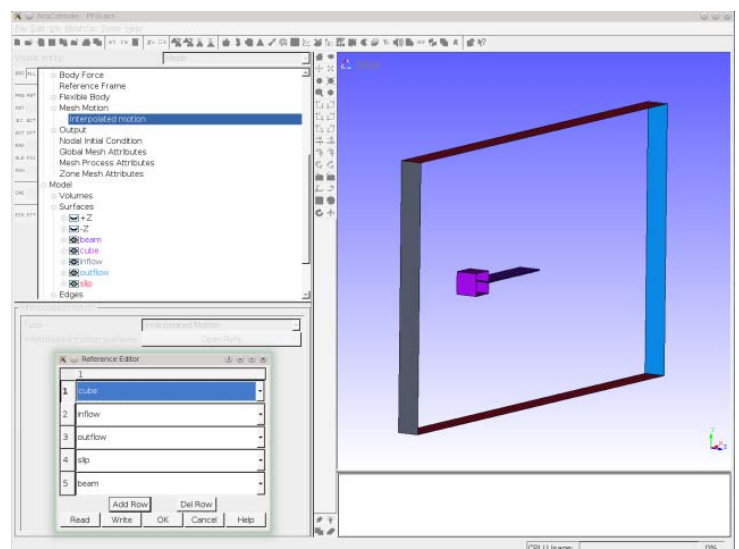
Difference in behavior between default ALE mesh motion (left hand image) and that achieved by setting the `mesh_volume_absorption_factor=3.5`.

Interpolated Mesh Motion

A new method of defining mesh motion within AcuSolve, known as "Interpolated Mesh Motion" has been added to the solver in 13.0. Interpolated mesh motion provides a shortcut for quickly defining simple mesh motions without the need to solve for the ALE equations. This reduces compute time for some moving mesh applications by as much as 50%. Interpolated mesh motion utilizes a wall distance based interpolation scheme to determine the motion of each node. This avoids the need to solve a hyper-elasticity equation to determine the mesh deformation and greatly increases the speed of the simulation. Interpolated mesh motion requires the definition of surfaces that form the boundaries from which the interior motion is derived.

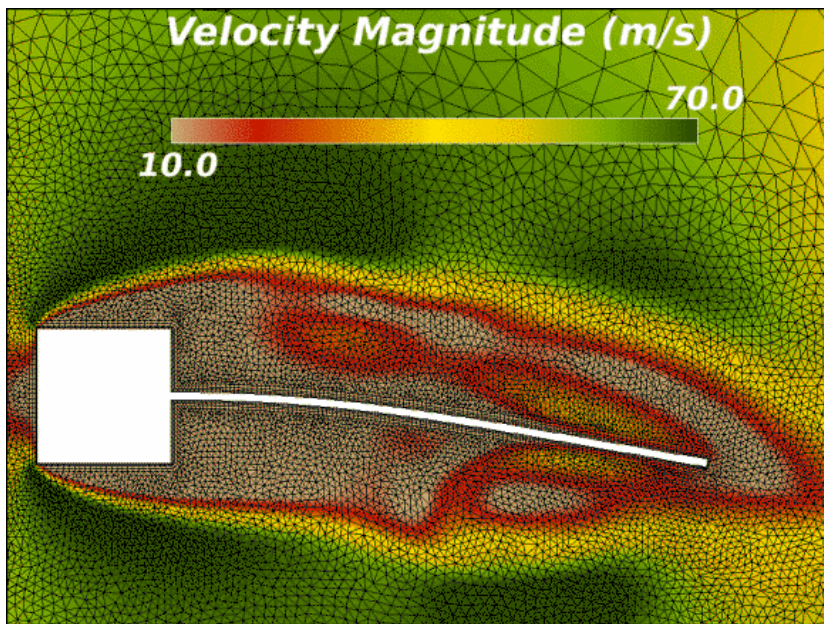
For example, to simulate a deformable cantilevered bar in the wake of a bluff body, the user would identify the wetted surfaces of the fluid volume as the interpolation surfaces. A mesh motion of `type=interpolated` would then be assigned to all nodes within the domain. As the bar deforms, the motion of the deformed shape will be distributed throughout the bounds of the region for which the interpolated motion is applied. In this example, the motion is distributed through the entire domain. However, internal surfaces could be used to limit the motion to only a portion of the volume if desired.

This feature may be used in combination with the ALE solver and on subsets of nodes to provide a powerful and flexible new technology for controlling the motion of the mesh. Note that the pre-processing expense of this feature scales with the total number of nodes on the collection of surfaces that define the bounding volume, the number of nodes to which the interpolation is applied, and the anisotropy of the domain.



Identification of bounding surfaces in AcuConsole for use with interpolated mesh motion. Note that in the case of a 2-d analysis, it is not necessary to select the 2-d planes.

Although this feature provides a powerful new technology, it is not intended to replace the ALE solver as a general purpose mesh motion solution for complex motions and geometries. Furthermore, it requires that all three components of the mesh displacement be fully specified on all INTERPOLATED_MOTION surfaces. This prevents it from being used on free surfaces and guide surfaces.



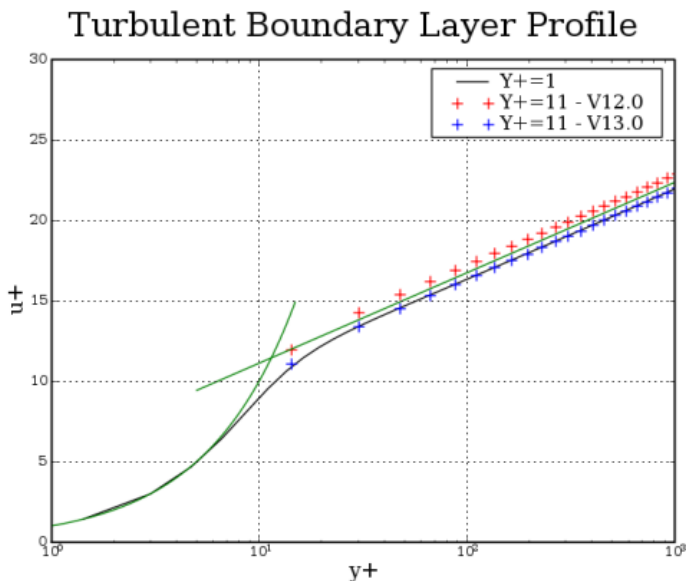
Animation showing the deformation of the mesh for a flexible bar in the wake of a bluff body.

New Smooth/Rough Wall Function for Spalart-Allmaras Based Turbulence Models

AcuSolve's wall function for Spalart-Allmaras based models has been modified to provide better agreement with the fully wall resolved solution. The newly implemented wall function retains the functionality of the previous technology and is valid through the full range of Y^+ values (0-~300). The new wall function offers improved agreement with the fully wall resolved solution, particularly when the first node off of the wall falls within the buffer layer. Legacy wall function behavior can be recovered by setting the following cnf parameter:

```
wall_function_spalding=off
```

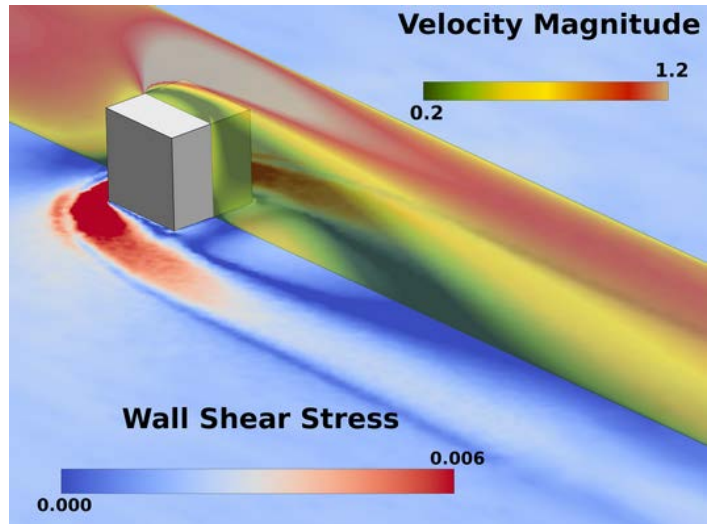
The new wall function implementation also allows users to specify surface roughness for any y^+ value. This removes the restriction in previous releases of AcuSolve that the first layer height must be greater than the wall roughness.



Boundary layer profile using new wall function for Spalart-Allmaras model.

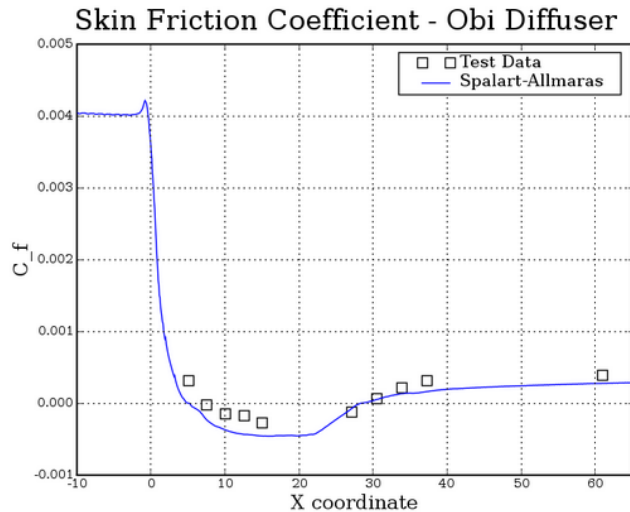
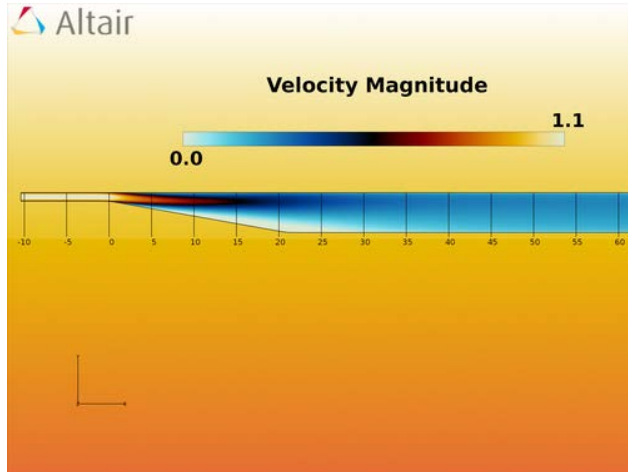
Wall Shear Stress Added to NODAL_OUTPUT and SURFACE_OUTPUT

Wall shear stress is a commonly requested output variable from CFD results. This quantity represents the viscous stresses acting on a wall and is often used for the purpose of comparing with experimental data and also to aid in visualizing flow patterns near the wall. In previous releases of AcuSolve, this quantity was computed in AcuFieldView as a post-processing task. Starting in 13.0, the wall_shear_stress variable has been added to the NODAL_OUTPUT and SURFACE_OUTPUT variables. The wall shear stress is computed based on the turbulent "Law of the Wall" and is available for any wall that has a no-slip condition applied to it. The wall shear stress magnitude is evaluated from the law of the wall, then the velocity at the first point off the wall is used to assign a vector direction to it. This allows users to visualize the magnitude and direction of the wall shear.



Wall shear stress magnitude used to visualize the flow patterns in the wake of a surface mounted cube.

Note that the approach used to compute the wall shear stress described above is not the same as what is used when computing the surface traction. The surface tractions are evaluated directly based on AcuSolve's finite element formulation. Small differences are expected if comparing the results from the two approaches.

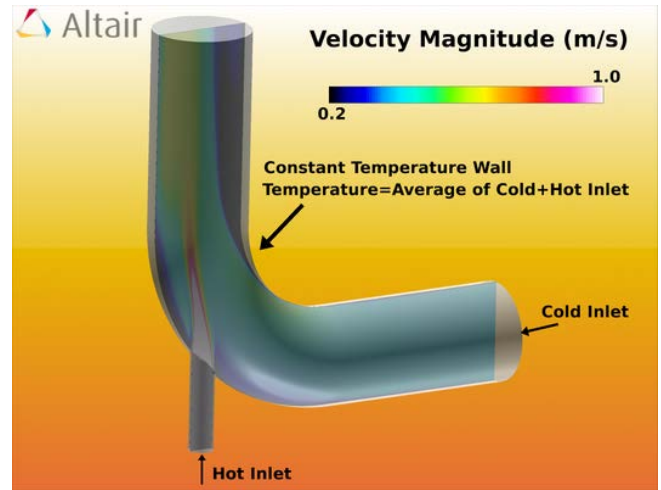


Wall shear stress used to compute skin friction coefficient for purpose of comparison to experimental data.

New options for modeling Rough Wall Heat Transfer

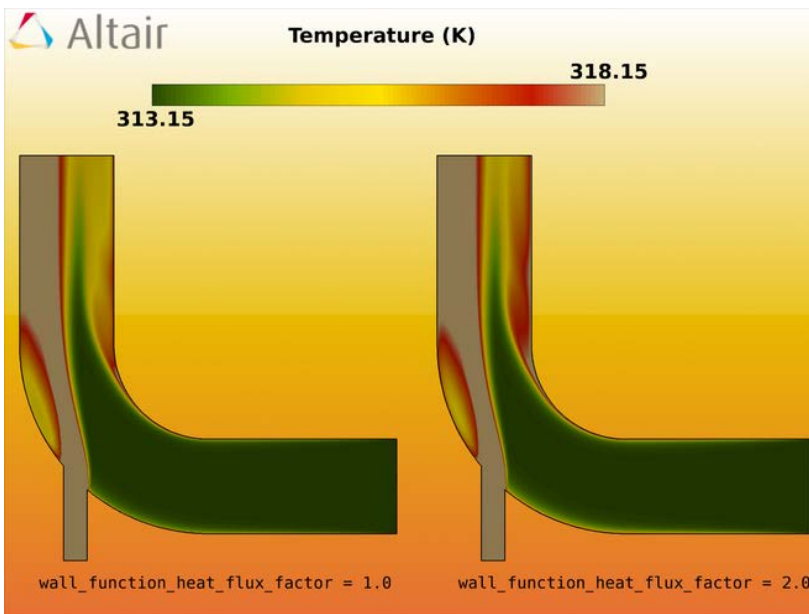
AcuSolve currently supports modeling of flow over surfaces containing smooth and rough walls. When the surface is not smooth, the roughness is incorporated into the momentum equations by modifying the viscosity near the wall. This causes a change in the near wall momentum and results in a shift in the boundary layer profile. When investigating heat transfer over rough surfaces, the thermal boundary layer is also impacted by the roughness. A new parameter has been introduced into AcuSolve 13.0 to account for this effect. The newly introduced

`wall_function_heat_flux_factor` is used to scale the turbulent thermal conductivity in the first element off the wall. The impact of this option is to scale the heat flux at the wall in a nearly linear fashion. This option can be loosely interpreted as the ratio of the Nusselt Number for a rough wall to that of the Nusselt Number for a smooth wall.



Mixing elbow heat transfer simulation in which the outer walls are held at a constant temperature and the 2 inlets are at different temperatures.

Comparison of the temperature field resulting from two different settings of the `wall_function_heat_flux_factor` reveals the impact of this parameter on the heat transfer.

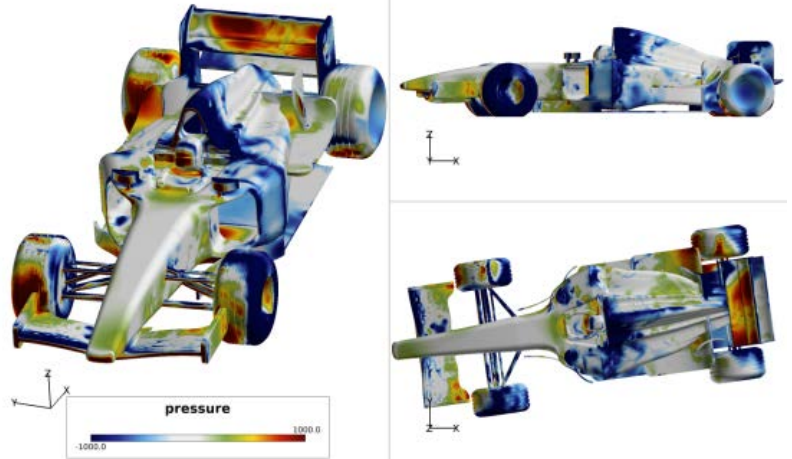


Comparison of results showing the impact of the `wall_function_heat_flux_factor`

Note the higher temperature near the outlet of the model due to higher heat transfer rates from the surrounding wall.

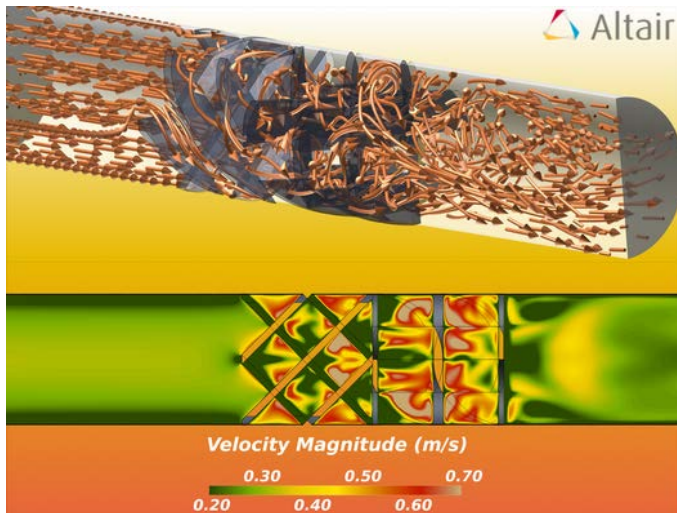
Improved CFD Post-processing With AcuFieldView

AcuSolve 13.0 comes equipped with a new version of AcuFieldView that is based on Intelligent Light's most recent release of FieldView. This release of AcuFieldView provides significant new post-processing capabilities to AcuSolve users. Among the new features are multi-window capabilities that enable users to simultaneously visualize different datasets within the same session. Users can now fully duplicate the content of a given window through the use of a single mouse click. The copy of the window can be independent and modifiable or linked to its parent window. All previously generated restart files are supported for use with the multi-window capabilities and FVX automation is fully supported within the multi-window environment.



Multiple images of the same model produced using AcuFieldView's multi-window capability

AcuFieldView comes equipped with enhanced capabilities for working with vectors. Starting with this release, users have the option of drawing "traditional" straight vectors or vectors that curve to follow the path of the flow. The curved vector option reflects the actual path of the flow throughout the length of the vector. Additional vector option improvements include a new method of reducing the number of vectors in an image by skipping over elements based on a simple masking pattern. This feature is an alternative approach to the current uniform sampling option that is present within AcuFieldView.

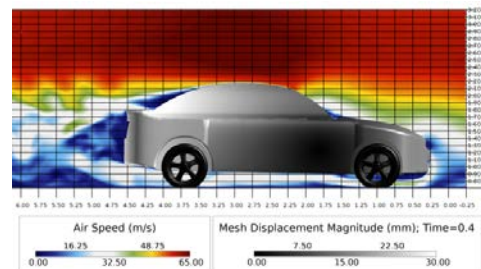


Curved vector and multi-window support

AcuFieldView also supports ruled grid display on coordinate surfaces. This option allows the overlay of a grid to illustrate scale of the objects in the display windows. The increments used on the grids, grid labels, and grid line color can be changed independent of the surface it is rendered on.

AcuFieldView now comes equipped with the DataGuide feature enabled. When activated, AcuFieldView traverses the dataset and writes pre-processing information regarding the data to disk. The time for subsequent read operations of the data are then reduced by 2-25x when these files are present. Note that there is a penalty in time and disk space necessary to create the DataGuide files. However, the benefit far outweighs the cost if the data is going to be visited more than once.

DataGuide can be activated by passing the -p2 option to AcuFieldView's launch command. Note that this option is typically used as an alternative to using parallel operation. It should also be noted that this feature is not available for transient datasets when the data is imported through the AcuSolve direct reader. Conversion of the data into FV-UNS format using AcuTrans or AcuOut is the best method of leveraging this new feature.



Coordinate surface showing the use of ruled grids.

Other notable changes

AcuSolve 13.0 contains a number of other notable changes that are worthy of mention. A brief description of each is shown below:

- Improvements to organization of time history output sets in AcuProbe
- Added SURFACE_OUTPUT variable `partial_volume`. When the surface output represents a closed set of surfaces, this quantity is equal to the volume of the surface. For partially enclosed surfaces, it has no meaning.
- Added `udfGetResidualNorm`, `udfGetResidualRatio`, `udfGetSolutionNorm`, and `udfGetSolutionRatio` to the list of User Defined Function calls. These functions can be used to compute the current level of convergence from within a UDF.
- Added the ability to specify momentum source terms on a per element basis using the `element_volume_momentum_sources` parameter of the ELEMENT_SET command.
- Added the ability to set the precedence of the `mesh_motion` parameter when applied to an element set. This option is used to control whether the `mesh_motion` applied to the element set should override any mesh motion constraints applied to the attached surfaces.
- Added the ability to specify the coupling direction and coupling type for external code surfaces coupled to rigid bodies.
- Added support for wall roughness with SST, k-omega, and SST-DES turbulence models.
- Renamed AcuRadtherm to AcuTherm and added the following features:
 - Ability to export results in CSV format to support import of data into JMAG
 - Added the option to compute heat transfer coefficients based on direct sampling of reference temperature, a user specified constant reference temperature, or the turbulent Law of the Wall.
- Expanded support for rough wall modeling to include low Reynolds Number walls and also removed the restriction that the first layer height must be greater than the roughness height.
- Added an option of `num_threads=_auto` to all parallel launch scripts to automatically set the optimal thread count for parallel simulations. This option interrogates the motherboard to determine the number of cores available and uses openMP for message passing between them. For jobs that span nodes of a cluster, distributed message passing will be used between nodes.
- Added an option of `max_ddc_agglomeration = auto` to AcuPrep to automatically agglomerate the mesh for domain decomposition purposes when the size exceeds 100 million elements. Note that this option can be disabled entirely by setting the value to 1.
- Added all necessary environment variables into AcuRun and AcuRunTrace to allow executing these programs from outside of the AcuSolve cmd window (Windows platforms) or without sourcing the `acusim.sh` file (Linux).
- Added a warning to AcuSolve that notifies users if a given motherboard is overloaded with processes.
- AcuFieldView now loads the last time step available on disk when launched from AcuConsole.
- Added the ability to specify additional run time options to AcuRun from the "Launch AcuSolve" panel in AcuConsole.
- Added support for higher order elements in AcuPev.
- Shortened the names of the surface statistics output variables to be `min_*`, `max_*` and `std_*`.
 - Note that this does break backward compatibility for AcuProbe UDF and automated post-processing scripts. Users should update their scripts to reflect the new naming convention.
- Shortened the names of the `mass_average_*` and `mass_flux_average_*` SURFACE_OUTPUT and ELEMENT_OUTPUT variable names.
 - Note that this does break backward compatibility for AcuProbe UDF and automated post-processing scripts. Users should update their scripts to reflect the new naming convention.

- Added support for reading ProEngineer Creo files into AcuConsole.

Resolved issues

- Resolved a problem with SURFACE_OUTPUT values on mixed element surfaces
- Resolved a problem with AcuTrace taking excessively long to initialize on large meshes.
- Resolved a crash of AcuRadtherm (now Acutherm) on WIN64 platforms
- Resolved a problem with ELEMENT_OUTPUT values for the eddy_viscosity and species variables
- Resolved a bug in AcuTrans that led to incorrect Enight files when using -ale and -rmds flags simultaneously
- Resolved an issue regarding stability of FSI applications depending on the order in which the mesh displacement and flow staggers are solved.
- Resolved an issue with the mesh_distortion_tolerance and mesh_distortion_correction_factor that prevented it from functioning properly.
- Resolved a crash of AcuFieldView on Win64 when using the AcuSolve direct reader and the output_initial_condition option of NODAL_OUTPUT was turned on.

Changes to supported platforms and third party packages

- 32 bit Windows platforms are no longer supported
- mvapich message passing is no longer officially supported on any platform, and will be completely removed from the distribution in the next release.

Changes to default settings

- The collect_subdomain_output option of AcuSolve is now turned on by default to minimize the number of files written to disk during simulations. For cases in which a large number of time steps are retained, this option greatly improves the response time of the file system, which improves the speed of post-processing operations such as AcuTrans, AcuFwh, and AcuTrace. It does, however, come at the expense of slightly longer output times during the AcuSolve run.
- The default setting for num_threads is now _auto for AcuRun and AcuRunTrace.
- The default setting for max_ddc_agglomeration is now _auto in AcuPrep and AcuDdc.
- The -ale option of AcuTrans is now turned on by default.
- Advanced CNF options for all compiled programs are now hidden and removed from the documentation.

Summary of new/changed AcuSolve input file commands

The following table summarizes the changed or newly introduced AcuSolve input file commands. Note that a full description of each command is available in the AcuSolve Command Reference Manual.

AcuSolve Command Name	Description
MESH_MOTION	Added type=interpolated_motion and interpolated_motion_surfaces to enable interpolation of motion in interior nodes based on the motion of bounding surfaces.
INTERPOLATED_MOTION_SURFACE	New command that was added to identify surfaces that will participate in a MESH_MOTION command of type=interpolated.

ELEMENT_SET	Added <code>element_volume_momentum_sources</code> to enable specification of body forces on a per element basis.
ELEMENT_SET	Added <code>mesh_motion_precedence</code> to control the resolution of conflicting mesh motion boundary conditions at surfaces connected to the volume.
NODAL_BOUNDARY_CONDITION	Added <code>coupling_type</code> and <code>rigid_body_name</code> to support coupling with MotionSolve.
SIMPLE_BOUNDARY_CONDITION	Added <code>wall_function_heat_flux_factor</code> for modeling turbulent heat transfer on rough walls.
EXTERNAL_CODE_SURFACE	Added <code>coupling_type</code> , <code>coupling_direction</code> , and <code>rigid_body_name</code> to support coupling with MotionSolve.
CAA_OUTPUT	Added <code>output_si_na_pressure</code> .

Summary of new/changed AcuSolve command line options

Program	Option	Description
AcuRun AcuRunTrace	<code>-num_threads</code>	Added a keyword of <code>_auto</code> that automatically sets the number of threads based on the CPU present on the machine and host list
AcuPrep AcuDdc	<code>-max_ddc_agglomeration</code>	Added a keyword of <code>_auto</code> that automatically sets the agglomeration to ensure that the maximum number of elements passed to the domain decomposition utilities is less than 100 million.
AcuSolve	<code>-wall_function_spalding</code>	Option that controls the wall function for the Spalart-Allmaras based turbulence models. Turning this option off reverts back to pre 13.0 behavior.
AcuSolve	<code>-mesh_volume_absorption_factor</code>	Parameter that controls the volume weighted stiffness of elements when performing simulations utilizing ALE.
AcuTherm	<code>-num_filter_passes</code>	Number of filter passes used to filter the film coefficient and reference temperatures.
AcuTherm	<code>-film_calculation_type</code>	Option used to select the method of computing the surface film coefficient.
AcuTherm	<code>-thermal_layer_edge_y_plus</code>	Option used to determine the distance from the wall in which to sample the reference temperature.
AcuTherm	<code>-user_reference_temperature</code>	Option used to define the constant reference temperature used in the calculation of the film coefficient.

AcuTherm	-translate_to	Option used to determine the output format of AcuTherm (Radtherm or JMAG).
----------	---------------	--

HyperXtrude Solver

There have been major performance enhancements for all HyperXtrude solvers in this release. Solvers can now handle large problems (up to 4 million nodes)

All Solvers

Performance Enhancements

HyperXtrude solvers can now handle large models on both Windows and Linux 64-bit platforms. In the previous release, size was limited to 450,000 nodes on Windows machines. Now the solver can handle up to 4 million nodes on both the platforms. In addition, there is a significant speed improvement.

HyperXtrude for Metal Extrusion

Improvements to Solver Accuracy

Accuracy of the nose cone prediction of profiles, especially those with large variation in thicknesses, is improved.

Changes to Bearing Optimization Module

Bearing optimization will determine the imbalance in material flow at the end of bearing region instead of the exit. This improves the accuracy of the bearing optimization.

Mandrel Offset

The recommended procedure is to flush the mandrel offset before creating the mesh. This is accurate and will also result in good quality mesh in the welding chamber region. In this release, you can also mesh the model with the mandrel offset. The solver will account for this step in the computation and the computed solution will be equally accurate. The advantage of including the mandrel offset is savings in time spent in CAD correction; however, there may be a disadvantage in creating good quality mesh in this region.

Grain Size Computations

You can compute grain size using UDF function or a TCL command. If these are not used, the solver was using a simple sub-grain size model. In this release, it has been replaced by a more accurate dynamic recrystallization (DRX) grain size model. This new model takes into account of dependence of grain size on strain and temperature corrected strain rate.

Resolved issues

- Error in the units printed for the surface area of boundary faces in the output balance tables are now corrected.
- Bug fixes to the bearing friction module have been made. The changes made in 12.0.210 resulted in increasing the stick zone in the bearing region for some models. This has been corrected. In addition, this module also has been updated to improve the load curve prediction for non-hexahedral mesh models.

- Bugs in exporting results in H3D file format have been corrected.

HyperXtrude for Billet Forging

Grain size computations

The solver now supports computation of grain size using static, dynamic, and metadynamic recrystallization models.

Material models

Flow stress of the workpiece can now be computed using the following constitutive models.

- Johnson-Cook
- Norton-Hoff
- Hensel-Spittel
- Flow stress data in 3D table format (as function of strain rate, strain, temperature)

In addition, you have the choice of using the existing Sellars-Tegart and metal forming power law models.

HyperXtrude for Resin Transfer Molding

Local Coordinate System Enhancements

The solver now supports local coordinate systems (cylindrical coordinates) for permeability and thermal conductivity tensors. In addition, exported H3D results now include the permeability and conductivity tensors to help visualize the local coordinate system data used in the model. The axis of the local cylindrical coordinate system can be X, Y or Z axis.

Automatic Time Step

The solver will adjust the time step based on the rate of filling. Hence, the solution can be started with a small time step (0.001 s) and the solver will handle the rest.

Mass Conservation

The filling algorithm has been improved to conserve mass. Also, the simulation has been improved result in sharper fronts.

Flow Rate Injection Boundary Condition

In addition to filling by specifying inlet pressure at gates, you can also specify volume flow rate and simulate filling.

Resolved Issues

- Bugs in resin front calculation are now resolved.

HyperXtrude for Metal Extrusion Process Management

HX-Process is software for extrusion process management and is used in synergy with HyperXtrude for metal extrusion. HX-Process comes in two flavors: Server edition and Personal edition. Server edition supports multiple users and enables them to access/work from a common database. This is suited for a company-wide implementation. On the other hand, personal edition is a single user version and works with the database tied to that user and the machine. Starting 13.0, HX-Process (personal edition) is packaged along with HyperWorks. Key features of HX-Process include:

- **Compute Optimal Process Parameters:** HX-Process can determine optimal values of ram speed, billet preheat temperature, and axial billet taper temperature based on extrusion limit diagram for the select profile, press, die, and material data.
- **Minimize Scrap:** HX-Process can estimate the optimal billet to minimize scrap.
- **Prepare Quotations:** HX-Process can estimate the costs and prepare quotation. In the personal edition, templates used for computation and report generation cannot be customized.
- **Manage Data:** HX-Process has databases for extruded profiles, presses available in the plant, dies used for extrusion, billet, and cost database. These data are used in computation and quotation preparation.

BasicFEA 13.0 Release Notes

13.0 Release Notes

Drag and Drop

Parts can now be dragged and dropped in and out of assemblies from the BasicFEA browser.

Localization

BasicFEA is now translated into Italian, which makes 13 total languages supported.

Maximizing Computing Power

Right before running an analysis, BasicFEA determines the number of usable cores and available RAM on the machine. This is then passed to the solver, and results in major runtime speedups.

Load Distribution Body

A new type of rigid body has been added. The **Load Distribution Body** is used to distribute applied loading without adding stiffness to the model.

Friction

Friction values can now be applied to all sliding pins and sliding contacts. Remember to use the Sliding Contact Loadstep to activate non-linear contact.

Preferences

All settings and preferences are collected in one location accessible at the top right side of the BasicFEA browser.

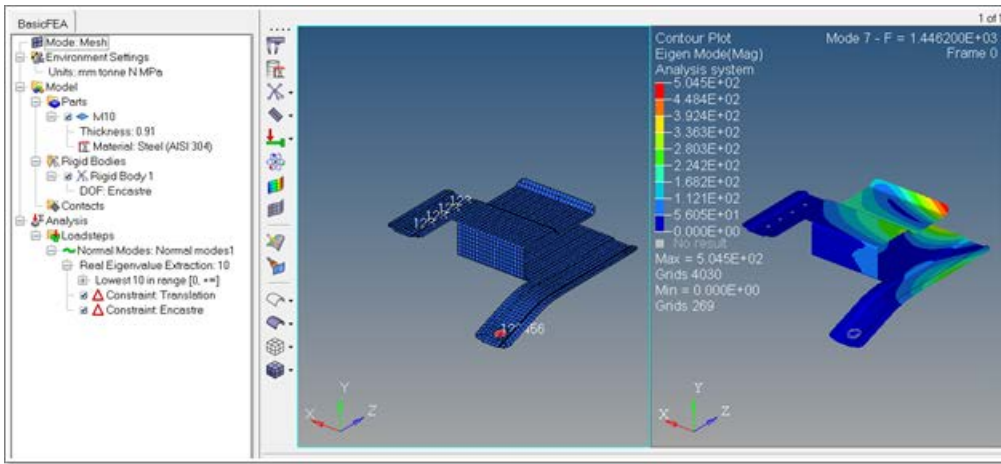
Resolved Issues

- User materials saved in units other than mm tonne N MPa were not being converted properly

12.0.110 Release Notes

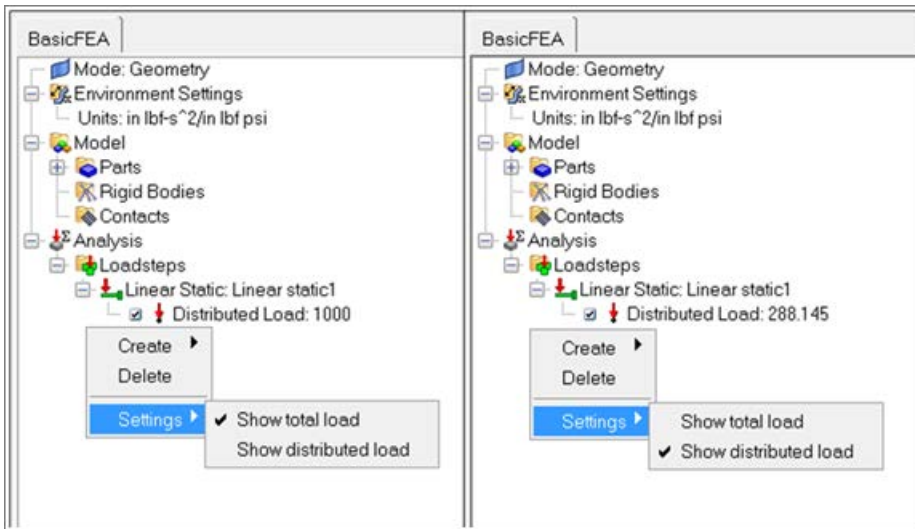
Shell Parts

BasicFEA now handles 2D surface geometry. Users are able to import 2D surface CAD, or midsurface their 3D solid CAD with the **Solid to Shell** tool available from the Parts menu in the BasicFEA browser. In order to support surface geometry, we've added shell meshing with the option for an optimized quality mesh. We've also added support for line selection with our Rigid bodies and Sliding pins. Finally, as long as you import 3D solid CAD and use our **Shell to Solid** tool, we offer the option to retrieve the original CAD with the **Shell to Solid** option.



Total Load Display Options

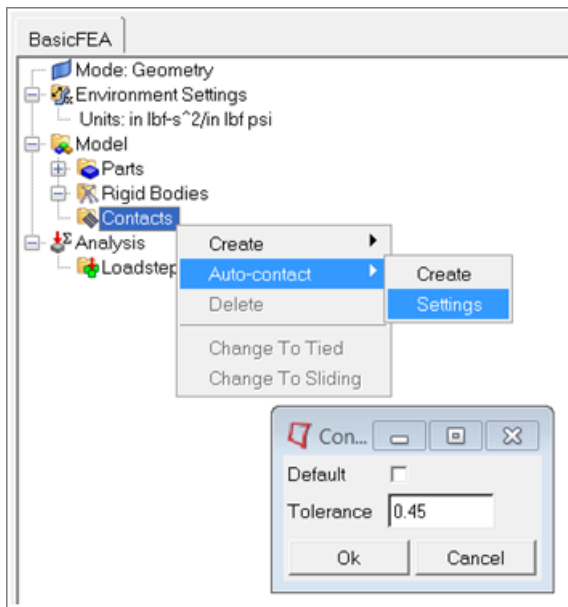
Under the Loadsteps right click menu, we've added a settings option to display the load value in the browser:



Show total load will display the total load applied to the selected surfaces, while **Show distributed load** will show the load applied per unit area (pressure).

Auto-Contact Tolerance Settings

The Auto-contact tolerance is automatically calculated based on the size of the imported model, but for more user control, we've exposed it in the right click menu under contacts.



Resolved Issues

- Fixed fatigue material conversion issue
- Added language updates

HyperWorks Collaboration Tools 13.0 Release Notes

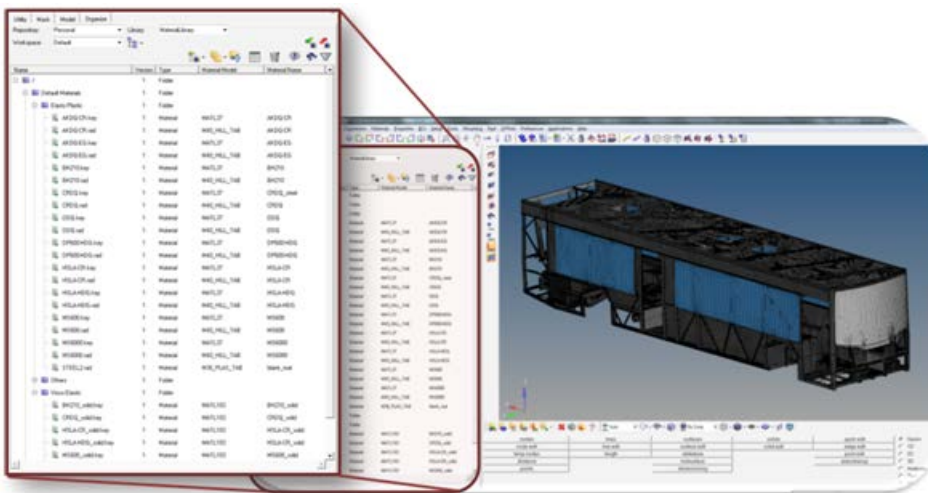
13.0 Release Notes

HyperWorks Material Library

Built on the Collaboration Tools framework, the Material Library is an out-of-the-box tool available with HyperWorks. It provides users with:

- rapid access to corporate or group material information needed by their simulations
- assurance that material properties used in simulations are up-to-date and accurate
- seamless integration into HyperMesh and the HyperWorks Desktop

Users no longer have to hunt around for required material information, and they no longer have to leave their pre-processor to find and seamlessly import such information.



Feature highlights include:

- Logical organization of material data and files
- Ability to add material data and files individually or in bulk
- Simple and advanced library searches, based on automatically extracted metadata and also text within material files themselves
- Library find and filter
- Preview and compare material properties or stress-strain curves
- View, edit, and full version control

Windchill Integration

With this new PDM connector, customers with PTC Windchill can now directly access CAD and other information stored in their corporate PDM system. Like the other PDM connectors, the Windchill connector allows users seamless log in and query within Windchill, without ever leaving the HyperWorks Desktop. The bi-directional connection allows users to both pull data and files (such as CAD geometry) from Windchill as well as push/publish (such as meshes, solver decks) back to Windchill.

Key to Metals Integration

With this new connector, Key to Metals customers can now directly access material data stored in their corporate material database. Like the other connectors, the Key to Metals connector allows users seamless log in and query within Key to Metals, without ever leaving the HyperWorks Desktop. Users can search for materials based on their Key to Metals metadata, including standards and language, and then select the mechanical and physical properties required for their simulations. The material information is exported from Key to Metals as the chosen RADIOSS material card type and can be directly imported into HyperMesh.

Delete/Obliterate Enhancements

Users can now select multiple contents to delete. It is now possible to obliterate all versions of a deleted content in a single step. Previously, contents had to be deleted individually, and versions also had to be obliterated one at a time.

Get To

Get to allows users to retrieve contents to a location other than their workspace. This is useful in cases where contents have very long folder paths in the repository, which, due to operating system limitations, cannot be resolved in the workspace.

When retrieving the contents, you have the option whether or not to retain the folder structure and whether or not to retrieve referenced contents.

Checkout Lite

Available only when using remote (team) libraries, **Checkout Lite** allows users to checkout a content without retrieving its document(s) to the workspace. This is helpful when only a metadata change is needed, and copying the document(s) to the workspace would be time consuming.

12.0.115 Release Notes

Resolved Issues

- Fixed the issue where files with German characters were not added correctly during the Add Folders and Files process.
- Resolved the issue with remote libraries where already defined workspaces were not visible.
- Resolved the issue where the application was moving to the background each time a new content was published.

HyperMath Solutions

HyperMath Solutions provide general mathematical tools for math scripting, modeling, study, and optimization.

HyperMath	Mathematical language for numerical computation
HyperStudy	Integrated optimization, DOE, and robust design engine

HyperMath 13.0 Release Notes

13.0 Release Notes

HyperMath-ScriptView Merger

With 13.0, ScriptView no longer exists as a separate product. All functions have been incorporated inside HyperMath. All ScriptView features are kept and the connection from HyperWorks Desktop products to execute or debug both HML and TCL scripts is maintained.

New Matrix Class

The underlying handling and storage of matrices has been redeveloped. The use of dedicated, standard libraries helps to provide better performance and a number of additional functions such as `eig()`, `svd()`, and improvements to existing functions.

Also, with this change, the handling of empty matrix, Inf, NaN and complex operations has been extended.

Path Management and File Browser

The handling of the current path has been revisited. The current path used in scripts is now in sync with (and can be modified from) the File Browser widget.

Library Browser and Console

The Library Browser has been re-implemented.

The console display has been improved (fonts, alignment). The completion mechanism has been revisited and made consistent between the console, the Library Browser and the Help pages.

Plot Enhancements

Besides resolving issues, the Plotting infrastructure has been redesigned. In addition new functions such as `AddPoints()` and `ClearPlot()` have been added.

Utility Functions

A number of utility functions have been added, such as: `what`, `who`, `which`, `ispc`, `isunix`, `fileparts`, `hmlfilename`, `isdir`, `unix`, `exist`, `time`, `tic`, `toc`, `pause`, and `memory`, along with some table management functions.

HyperStudy Function Registration

Similarly to HyperGraph in previous versions, HyperStudy functions can now be registered automatically in HyperStudy from within the HyperMath Editor window.

Resolved Issues

- Clear and memory issues
- Empty matrix handling issues
- OptiStruct-HyperMath bridge issues

12.0.110 Release Notes

Handling of ASCII File

The editor can now read in all ASCII files.

Memory Management

- Improvements to the memory management
- Memory allocation issues are caught and reported as error messages
- Memory is cleared and possible leaks have been tracked

Empty Matrices

A number of the Math functions have been improved to properly handle empty matrices.

Error Handling

- Improvements to the error handling and error messages
- Handling of error returns within the Eval() function
- In conjunction with the empty matrix handling, associated error messages are put in place or improved

Math Library Enhancements

- Numerous enhancements to equality and comparison tests
- Improved support of complex numbers in some functions

Resolved Issues

- Slowness of the Dir() command observed on some systems is solved
- Wrong Warning messages in batch mode are removed
- As a consequence of the deprecation of the local keyword, all scripts where local is used to define a variable are initializing those variables to nil
- Fixed plotting performance issues on Linux
- Fixed display issues related to the Legend on plots
- Improved quality and display of curves markers
- Fixed the returned value of plot handles when using PlotLine()

- Fixed text font and text color issue in the HML command window upon copy/paste actions
- Fixed issue with the HyperMath plugin in HyperView Results Math. Multiple HyperMath derived results can now be used.

HyperStudy 13.0 Release Notes

Introduction

HyperStudy enables users to explore, understand and improve their system designs using methods such as design-of-experiments and optimization. HyperStudy generates intelligent variations of the parameters of any system model and reveals relationships between these parameters and the system responses. Using HyperStudy, users can make better decisions and optimize the performance, reliability and robustness of their systems.

HyperStudy continues its fast pace of development based on the updated product in version 12.0. In HyperStudy 13.0, study setup is simplified via simplified model definitions and a new powerful parameter editor. Fitting results can now be reported to Excel spreadsheets allowing ease-of-use and portability. In HyperStudy 13.0, user interface can be displayed in five additional languages: Chinese, Japanese, Korean, German and French.

13.0 Release Notes

Excel Plug-in

Trade-off 1D post-processing of the Fit approach can now be reported to Excel spreadsheets, allowing trade-off studies in Excel, independent of HyperStudy. This makes the trade-off studies portable and usable. This capability withdraws 5 HyperWorks units. In order to use this functionality, users need to add the HstAddinFit plug-in to Excel the first time they are using it.

Improved HyperMesh and MotionView models

You can now add HyperMesh and MotionView as models directly in HyperStudy; there is no need to open the models in HyperMesh or MotionView first and start HyperStudy from within the pre-processor.

New Ansys Workbench Model

Ansys Workbench is a new model type in HyperStudy 13.0. Inputs and direct outputs of a Ansys Workbench project file (.wbproj) are imported as design variables and responses. This capability works with Ansys Workbench versions of 12.1 through 14.0 on win64, and win32.

New Parameter Editor

Editor is replaced by a new Parameter Editor. Parameter Editor performs better with large files. It also has a streamlined but powerful user interface.

HyperMesh Parametrization

In HyperMesh 13.0, any numeric attribute can be made a parameter in the LS-DYNA user profile. These parameters can then be imported to HyperStudy to be used as design variables.

Other Enhancements

- **Purge:** Purge allows you to minimize the size of your study directory during and/or after an evaluation by removing unnecessary run files.
- **Localization:** HyperStudy user interface can now be displayed in five additional languages: Japanese, Korean, Chinese, German, and French.
- **Sweep study:** Sweep study is a new specification in study setup. It allows the user to quickly create a sweeping type study.
- **Boxplot:** Box plot is a graphical representation of data that displays the distribution of data. It displays the spread and skewness of the data and easily identifies outliers.

Resolved Issues

- `hst_ng`, `hstbatch_ng` executables are retired.
- HyperStudy no longer hangs on importing variables when a WebEx application is open.
- In the DOE post-processing, interactions panel, a normalization problem that caused incorrect y-values is solved.
- In stochastic approach, initial values for the design variables can now be changed.
- When using an external optimizer HyperStudy did not write VALIDAT parameter in the `.opt` file. which caused the optimizer to start evaluation in the validation step. This no longer is an issue.
- Given a run matrix, the returned linear regression was incorrect due to the wrong scaling factor used in some cases when negative numbers were used. This no longer is an issue.
- DV initial value was reset to lower bound when values cell under details tab is opened, even if it is canceled. This no longer is an issue.
- If the design variables of a fit are linked to other design variables, fits did not return the correct values for the responses. This is no longer an issue.
- The `resvec` GUI did not display the correct contents; each data was 1 index off from the actual position. This is no longer an issue.
- Performance issue when extracting from large output files is solved.

Known Issues

- An aborted connection between HyperStudy and either HyperMesh or MotionView models cannot be re-connected within the currently open HyperStudy session. In order to re-establish the connection, HyperStudy must be restarted.

12.0.110 Release Notes

Setup: Simplified Study Setup by Auto Fill

In Add Model step, when an unparametrized file is picked as a resource, the filename is automatically written as input and the file is opened in the Editor as a `.tpl` file.

Fit: Enhanced Post-Processing

Trade-off 1D page is enhanced to show the relative change in the responses.

Optimization: Simplified Specifications

- Only methods that are applicable to the problem formulation (design variable, objective and constraint properties) are displayed.
- Method parameters are divided into two as Basic and Advanced Settings.

Optimization: Enhanced Iteration History Table, Iteration 2D and Iterations Plots

- Iteration History indicates the individual constraint violations. Constraints that are violated are in bold font.
- Iteration Plot2D Objective plot indicates the infeasible designs. Infeasible designs are marked with a larger circle.
- Iteration Plot2D has an option to display bounds. When selected, design variable, response and constraint bounds are displayed with lines.
- Iterations chart in optimization post-processing has a channel selector for the iterations. Using this selector, Pareto Fronts can be displayed.

General Improvements

- **Setup:** MotionView enable combo box when MotionView system has more than one model
- **Setup:** Basic MS/Excel connection now supported on Mac
- **Optimization:** Optimpost is retired and its functionality is replaced with enhanced optimization table and plots (see *Optimization: Enhanced Iteration History Table, Iteration 2D and Iterations Plots* list above).
- **Optimization:** Triangular and uniform distributions can now be assigned to random design variables and random parameters SORA, SORA_ARSM and SLA methods.
- **Stochastic:** Design variable distributions can be censored by their upper and lower bounds.
- **General:** UTF-8 support is added for Labels, Comments, Notes.
- **General:** Drag and drop of archive files (.hstx file) to an existing HyperStudy application imports the archive file.
- **Help:** Help re-writing is complete. The new help contains all new content, including updated tutorials and an improved tutorial system.

Resolved Issues

- Duplicate labels will fail in the HTML report generator
- An excess of 254 variables will not display properly in the run matrix
- Issue: [273542] OPT: different ARSM results because of different GUI workflow
- Multi model use in conjunction with the Extract tool will not work properly
- Issue: [277262] resvector Builder - Time channel in ODB files
- With Multi model the Task Manager text updates are not synced with the write/execute/extract
- A success may show up but another write needs to be performed
- Following tutorial file is not up to date tutorials/hst/HS-4230/plate.tpl
- Default graphical difference tool is slow on large files. The expectation is user uses their own. (see tips/reminders)
- If testing a win32 installation on a win64 machine the radioss.bat and run_dss.bat files need to be modified for OS_BITS

Manufacturing Solutions

Manufacturing Solutions is a unified environment for manufacturing process simulation, analysis, and design optimization. Use the links below to access the release notes for the following applications:

HyperForm	A unique finite element based sheet metal forming simulation software solution
HyperXtrude	Provides custom tools and easy to use wizards for setting up models to analyze metal extrusion, polymer processing, and metal rolling applications using HyperXtrude solver.
HyperMold	Provides a highly efficient and customized environment for setting up models for injection molding simulation with Moldflow (Midplane, Fusion, and Flow3D) and Moldex3D (Shell and Solid).

HyperForm User Profile 13.0 Release Notes

13.0 Release Notes - Pre-processing

Common (RADIOSS and LS-DYNA)

- Multistage process setup using the User Process has been updated with a setup page for easy definition and quick overview of all the stages in the multistage forming process.
- Process optimization
 - Enhanced to handle optimization in multistage forming process
 - New objective function supported – Minimize springback
 - New design variable added – Blank shapes
- Custom animation definition is made available for every tool motion definition for ease of animation file request based on the press motion like in real stamping process.
- Trim line / Blank shape predictor tool introduced for robust prediction of Trim line/Blank shape using incremental simulations results for net shape manufacturing.
- Option to define stroke for binder as an additional input in the Auto process single action model setup for better positioning of punch relative to the binder/blank holder.
- Support of Datum information in the solver deck for repositioning upon loading them in HyperForm.

RADIOSS Only

- New RADIOSS Mode based on Advanced Mass Scaling (AMS) solver technology that improves solution speed and accuracy has been supported for both adaptive use case (Fast - Fine mesh) and non-adaptive use case (Fast [Adaptive] – new).
- Support for modeling Hot Stamping process using new material model (Hot stamping HS Steel) that accounts for mechanical, thermal and metallurgical changes during the forming process.
- Support for modeling composite forming process has been introduced through the Auto process. It supports composites modeling using sandwich and multilayered approach.
- Hydroforming model setup has been enhanced to account for axial feed in the process definition.
- Improvement in solver solution speed when the drawbeads are modeled with simplified line approach.
- LAW80 – Hot forming material law. This law allows modeling the ultra-high strength steel behavior at high temperatures and the phase transformation phenomena from austenite to ferrite, pearlite, bainite and martensite during cooling.
- FAIL/NXT – New stress based failure criteria implemented based on the Nxt Failure model.
- INTER/TYPE 21 – Metal forming contact
 - Supports non-uniform temperature distribution on the master tool surface.
 - Accounts for friction energy in the thermal calculations.
 - Supports modeling of tailor welded blank with thickness defined at /PART keyword.
 - Performance and scalability improvement.
- INTER/TYPE8 – Drawbead contact
 - Supports normal force and bead depth in the contact calculations.
- DT/THERM
 - Time step calculation based on thermal parameters to speed up thermal only simulations like heating and cooling.

Resolved Issues

- Common - Incremental RADIOSS and LS-DYNA
 - Tool sits far away from the blank (shell and solid) during initial positioning.
 - Unable to card edit the material after creation
 - Animation count in the User Process by default is set as off.
- Incremental - RADIOSS
 - State file request after run time
 - Multistage model setup has been improved for robustness. Several issues addressed include:
 - Tool takes blank element ID resulting in failure of solver run
 - Adaptivity reset between stages
 - Trimming failure
 - Reading STA file from previous stages
 - Tools passing through the stoppers
 - Solver fails to run due to improper connectivity because of trimming
 - Guide pin modeling has been improved with new contact that avoids common solver error of excessive penetration and failure of solver run
 - LAW 78: Stability issues
 - LAW 73: Engine stop when extrapolation from tables was giving yield stress less than zero
 - Unexpected large number of animation files written by engine
 - Slow starter performance with model having drawbead contact INTER/TYPE8
 - Compatibility of Barlat material model with Advanced Mass scaling
- Incremental - LS-DYNA
 - Fixed issue of missing local system definition in input deck for process setup that involves local tool motion
- Die Module
 - Extract the outer boundary of the part as trim line based on geometry
 - Performance of punch and binder creation from die has been significantly improved
 - Irregular addendum creation when the part is hidden
 - Random crash of the software while adding or removing ribs has been fixed
- OneStep RADIOSS
 - While plotting formability either from the OneStep browser or FLD Plots panel, mismatch of colors corresponding to different zones in Formability contour and FLD plot is fixed.
 - Difference in the initial blank shape exported in IGES and STEP format.
 - Renaming the Drawbead in OneStep tab changes the force level to high
- Blank Fit and Blank Nesting
 - The blank shape exported after blank fit is not in the component position.
- Results Initializer
 - Handling component associated with material names that have special characters

13.0 Release Notes - Post-processing (HyperView)

New Features/ Enhancements

- Drawbeads defined using simplified line model will be displayed in HyperView for efficient post processing.
- Support %thinning for blanks modeled with solid elements.

- New post processing tools added :
 - Distance between Objects – Calculate the gap between two objects.
 - Section Cuts – Allows the user to define section cut and plot the contour variation along the section cut as 2D graph in HyperGraph.

12.0110 Release Notes

Incremental

Common (RADIOSS and LS-DYNA)

- Support definition of user defined location to store user defined materials
- Support setup of forming process using tailor welded and multilayered blanks in the User Process
- Introduction of new option to request animation file as function of tool stroke for each moving tool in the User Process
- Option to define force as a linear curve added in the User Process
- Stroke has been added to the binder for relative positioning of tools in the Auto Process
- New user friendly browser introduced to setup process optimization problem and execute through HyperStudy
- Springback positioner tool introduced to position the springback part relative to the tool/part before springback as a precursor for springback comparison and compensation

RADIOSS Only

- Gravity setup improved for speed and accuracy
- Hydrobending setup added to support tube bending with fluid mandrel

LS-DYNA Only

- Support of thermal forming setup for hot stamping simulation

Resolved Issues

- Incremental – RADIOSS
 - Saving material with multiple curves in the material database
 - Local coordinate system definition using solid elements
 - Missing self contact definition for the blank in the solver deck
 - Renumbering of nodes in the springback setup after trimming
 - Mesh refinement after gravity in multistage run using fine mesh option
 - Explicit engine keywords in implicit gravity deck

Die Module

Resolved Issues

- Trim line extraction for part with symmetric boundary conditions
- Improper trim line definition while adding and removing ribs

RADIOSS One Step

Enhancements

- New user friendly browser introduced to setup process optimization problem and execute through HyperStudy
- Export of flattened blank geometry in STEP format

Resolved Issues

- Duplicate curves created when importing FLC curve

Blank Nesting

Enhancements

- Support for costing in blank nesting

Post-processing (HyperView)

New Features/ Enhancements

- Line tracing tool improved for accurate blank shape and trim line prediction/tracing

Resolved Issues

- Tool-blank macro fails to calculate the distance and results in distorted mesh

HyperXtrude User Profile 13.0 Release Notes

HyperXtrude/Metal Extrusion

Automatic Extraction of Flow Volume

A macro to automatically extract the negative volume for the tool assembly with just one-click has been added. This macro will greatly minimize the model preparation time.

Automatic Billet Creation

A macro to automatically create a billet is implemented in 13.0. This macro can determine the billet center from die surface or edge. In addition, you can directly specify the center coordinates. This macro will automatically estimate the upset billet length from the specified data.

Easy to Use Geometry Cleanup Macro

A macro to automatically perform geometry cleanup is implemented in 13.0. This macro will estimate the target, minimum, and maximum element sizes for the solids in the selected component and automatically clean the solids. Users will have the option to change the estimated sizes before cleanup.

Utility Menu Reorganization

The Utility menu has been reorganized to access all features easily and also to reflect process flow.

Specifying Billet Taper Data

Billets are often heated with temperatures varying axially from the dummy block end to the die end. This is called **Axial Taper**. Similarly, **Radial Taper** and **Lateral Taper** heating is also used. Using the new **Billet Taper** macro, you can specify these three types of tapers in the analysis.

- Axial taper specification can have up to five zones and temperature can vary linearly in each zone with different gradients.
- You can specify lateral taper and radial taper in the model. These do not vary by zones.
- This data is required only for transient analysis.

Starter Billet

Using the Billet Taper macro, you can specify starter billets of different lengths and use different ram speed for each billet. This data is relevant only for transient, multicycle analysis. You can specify up to 5 different initial cycles with varying billet length data. In addition, you can use different axial tapers for each of the initial cycles. Data specified for the last initial cycle will be used for rest of the analysis.

Super Alloy Extrusion Using Glass Lubrication

The Extrusion Wizard now supports glass lubricated (with glass pad and glass powder coating) extrusion. This is often used with nickel, titanium, and steel alloy extrusion. Previously released versions of the

HyperXtrude solver did not support the analysis of these extrusions and you will need a custom version of the solver.

Glass Pad Material Model

The user interface now supports a new material type, **Glass Pad**. This material data can use either polynomial or VFT constitutive laws. In addition, the interface can fit the coefficients for VFT model directly from the composition of the glass. In the material database, there is another material type called **Glass Properties**; this is provided for backward compatibility and will be removed in 13.0.110.

Mandrel Offset

While meshing the models with die mandrel, it is recommended to mesh a flushed model; that is, with no offset in the mandrel. This will also result in mesh with good quality elements in this region. However, in this release you can mesh the model with the offset. If the offset is included, then you should specify the length of the **Mandrel Offset** in the bearing BC data. This will enable the solver to account for this in bearing friction computations. Due to this change, data exported in 12.0.110 will require the 12.0.210 solver installation. This selection is enabled in the Export panel.

Tool Deflection Analysis in User Units

The Tool Deflection Wizard now support user units. For example, if you set up the model in British units, results from OptiStruct will have displacements in inches and stresses in ksi. This is accomplished by automatically scaling the material data to create unit consistency.

Easy to Use Solid Edit Macros

The macros **Connect** and **Merge** are added to enable one-click solid editing. Merge will merge the selected solids. Connect will create a shared common surface at the interface between adjacent selected solids.

Fit Material

The Fit Material interface in the HyperXtrude material database has been improved. Users can now specify the material data interactively or via the DAT file as in previous versions.

HX Job Manager

Support to submit OptiStruct runs on remote machines has been added.

Resolved Issues

The following critical bugs in the HX Job Manager are resolved:

- Issues related to automatically creating password free connection are resolved.
- Putty and Gzip locations can manage folder name with spaces. This will enable the user to install them in the Program Files folder.
- While setting up clients, user can pick BAT files instead of executables.

The following issues were resolved in the Tube Extrusion Wizard:

- Unsupported unit systems are disabled.
- Errors in geometry creation are resolved.
- Errors in specifying correct friction conditions are resolved.
- Time step is computed correctly even if the user does not use the variable time step option.

Resolved errors in estimating billet diameter in the Parameters panel; now the Billet-Container BC is used to estimate this value.

Sometimes the user specified mesh in **my_2D_Bearing** is deleted if the tet mesher fails. This issue is now resolved.

HyperXtrude for Polymer Processing

Automatic Extraction of Flow Volume

A macro to automatically extract the negative volume for the tool assembly with just one-click has been added. This macro will greatly minimize the model preparation time.

Easy to Use Geometry Cleanup Macro

A macro to automatically perform geometry cleanup is implemented in 13.0. This macro will estimate the target, minimum, and maximum element sizes for the solids in the selected component and automatically clean the solids. Users will have the option to change the estimated sizes before cleanup.

Utility Menu Reorganization

The Utility menu has been reorganized to access all features easily and also to reflect the process flow.

Tire Extrusion Wizard

A new model setup wizard to simulate extrusion of tire treads is implemented.

- Create either the geometry based on given data or use the user specified solids for each component
- Supports the extruder type **multi-cut**
- Specify the tread/die contours either as symmetric or full contours
- Store and reuse specified contours
- Includes a report generator

Improve Mesh by Remeshing

All the wizards under Polymer Processing now have the **Improve Mesh** option. This improves the quality of the mesh generated by remeshing the solids based on the initially generated mesh.

Tool Deflection Analysis Wizard

This model setup wizard allows you to solve tool deflection analysis using OptiStruct.

- Automatically meshes using linear or higher order tets.

- Supports pressure or force load boundary conditions.
 - Maps the loads from HX analysis.
 - Model contact conditions in the model.
 - Specify the temperature condition for each component and also specify appropriate constraints on boundary faces.
-

Resolved Issues

Some critical issues in the **Polymer Melt Flow Analysis Wizard** are resolved. Now the inlet and outlet can have any axial orientation, however, you should use tet mesh or solid map in this region (drag will not work and will require orientation along Z axis). In addition, issues in organizing the exit BCs are now resolved.

12.0.110 Release Notes - HyperXtrude/Metal Extrusion

Specifying Billet Taper Data

Billets are often heated with temperatures varying axially from the dummy block end to the die end. This is called **Axial Taper**. Similarly, **Radial Taper** and **Lateral Taper** heating is also used. Using the new **Billet Taper** macro, you can specify all these three types of tapers in the analysis.

- **Axial taper** specification can have up to five zones and temperature can vary linearly in each zone with different gradients.
- You can specify **lateral taper** and **radial taper** in the model. These do not vary by zones.

This data is required only for transient analysis.

Starter Billet

Using the **Billet Taper** macro, you can specify starter billets of different lengths and use different ram speeds for each billet. This data is relevant only for transient, multi-cycle analysis. You can specify up to 5 different initial cycles with varying billet length data. In addition, you can use different axial tapers for each of the initial cycles. Data specified for the last initial cycle will be used for the rest of the analysis.

Super Alloy Extrusion Using Glass Lubrication

The Extrusion Wizard now supports glass lubricated (with glass pad and glass powder coating) extrusion. This is often used with nickel, titanium, and steel alloy extrusion. Previously released versions of the HyperXtrude solver does not support the analysis of these extrusions and requires a custom version of the solver to be installed.

Glass Pad Material Model

The user interface now supports a new material type, **Glass Pad**. This material data can use either polynomial or VFT constitutive laws. In addition, the interface can fit the coefficients for VFT model directly from the composition of the glass. In material database, there is another material type called **Glass Properties**; this is provided for backward compatibility and will be removed in 13.0.

Mandrel Offset

While meshing the models with die mandrel, it is recommended to mesh a flushed model; that is, with no offset in the mandrel. This will result in mesh with good quality elements in this region. However, in this release you can mesh the model with the offset. If the offset is included, then you should specify the length of the Mandrel Offset in the bearing BC data. This will enable the solver to account for this in bearing friction computations. Due to this change, data exported in 12.0.110 will require the 12.0.210 solver. This selection is enabled in the Export panel.

Resolved Issues

The following critical bugs in the HX Job Manager are resolved:

- Issues related to automatically creating password free connection are resolved.
- Putty and Gzip locations can folder name with spaces. This will enable the user to install them in "Program Files" folder.
- While setting up clients, user can pick BAT files instead of executables.

The following issues were resolved in the Tube Extrusion Wizard:

- Unsupported unit systems are disabled
- Errors in geometry creation are resolved
- Errors in specifying correct friction conditions are resolved
- Time step is computed correctly even if user does not use the variable time step option

Other resolved issues:

- Resolved errors in estimating billet diameter in Parameters panel; now Billet-Container BC is used to estimate this value.
- Sometimes the user specified mesh in "my_2D_Bearing" is deleted if tet mesher fails. This issue is now resolved.

12.0.110 Release Notes - HyperXtrude for Polymer Processing

Tire Extrusion Wizard

A new model setup wizard to simulate extrusion of tire treads is implemented.

- This wizard can either create the geometry based on given data or use the user specified solids for each component.
- In this release, the extruder type **multi-cut** is supported
- You can specify the tread/die contours either as symmetric or full contours
- Specified contours can be stored and reused
- This module also comes with a report generator

Improve Mesh by Remeshing

All the wizards under Polymer Processing now have the **Improve Mesh** option. This improves the quality of the mesh generated by remeshing the solids based on the initially generated mesh.

Tool Deflection Analysis Wizard

This model setup wizard allows you to solve tool deflection analysis using OptiStruct.

- It automatically meshes using linear or higher order tets
- Supports pressure or force load boundary conditions
- You can map the loads from HX analysis
- You can model contact conditions in the model
- You can specify the temperature condition for each component and also specify appropriate constraints on boundary faces

Resolved Issues

Some critical issues in the Polymer Melt Flow Analysis Wizard are resolved. Now the inlet and outlet can have any axial orientation, however, you should use tet mesh or solid map in this region (drag will not work and will require orientation along Z axis). In addition, issues in organizing the exit BCs are now resolved.

HyperMold User Profile 13.0 Release Notes

Moldflow

Solid Meshing Wizard

You can now re-mesh the model inside the wizard. This feature will not allow selective remeshing and mesh all the solids selected for meshing by the wizard.

Midplane Wizard

Enhanced with single click operation to set or extract thickness operation for all surfaces.

Property Cards for Hot and Cold Runner 3D

Interface will now create property cards for hot and cold runner 3D elements.

Thickness Extraction

Users can now ignore eccentricity while extracting thickness. This will set the value of eccentricity to zero.

Resolved Issues

- All solid elements created by the wizard will be split and converted to linear tetrahedral elements. Moldflow supports only linear tetrahedral elements.
- Performance of 2D surface meshing has been improved.
- Improved Solid Meshing Wizard's CFD tetmeshing module to handle a single solid with inlet and outlet.
- Melt entrance of the runner system cannot be at the moldbase exterior surface and it is always inside. The interface has been updated to handle this modeling requirement.

Moldex3D Solid

Solid Meshing Wizard

User can now remesh the model inside the wizard. This feature will not allow selective remeshing and mesh all the solids selected for meshing by the wizard.

Resolved Issues

- Diagnostics does not support hexahedral elements. If model had hexahedral elements, it was throwing application error and it is resolved now.
- Few issues were causing the graphics area to freeze and they have been resolved.
- Improved solid meshing wizard's CFD tetmeshing module to handle a single solid with inlet and outlet.

Moldex3D Shell

Importing Data Decks

This feature has been enhanced to include the BC data in import.

Midplane Wizard

Enhanced with single click operation to set or extract thickness operation for all surfaces.

Resolved Issues

- Users can now select the export folder location
- Few issues in the Midplane Wizard have been resolved. These include:
 - Graphics area freezing while creating property cards
 - HyperMesh hanging while creating feed system with trapezoidal cross-section
 - Some popup application errors while assigning thickness, etc.
 - Errors in redesigning feed system

HyperMold - HX/RTM

Pressure Data in Table Format

Users can specify the gate pressure a time dependent function in a table format. The interface supports both constant value and table specification.

Permeability Data in Local Coordinate System

Permeability tensor can be specified in using local coordinate system. The interface supports cartesian, cylindrical, and spherical coordinate systems.

Resolved Issues

- Improved solid meshing wizard's CFD tetmeshing module to handle a single solid with inlet and outlet.

Engineering Solutions

HyperWorks Engineering Solutions is a customized and highly automated environment for modeling and visualization environment of NVH, CFD, Crash, Durability, Suspension and Drop Test applications.

CFD	High quality tools for CFD applications enabling the engineer to perform modeling, optimization and post-processing tasks efficiently.
NVH	HyperWorks environment customized for automotive full vehicle NVH modeling and analysis needs.
Crash	A tailored environment in HyperWorks that efficiently steers the Crash CAE specialist in CAE model building, starting from CAD geometry and finishing with a runnable solver deck in both solvers RADIOSS and LS-DYNA.
Aerospace	A new user profile was added in the 12.0.110 release. This profile is aimed to help to capture the aerospace model building process and tools necessary for process automation.
Drop Test	An automated solution that allows you to either simulate a single drop test or a choice of multiple iterations with the aim of finding the sensitivity of process variables like initial orientation and drop height in a typical drop test by controlling the run parameters and conditions with ease.
Durability Director	Solver-neutral, process-oriented customization of HyperWorks that addresses many of the challenges associated with assessing the fatigue life of mechanical components.
Suspension Director	Industry specific solution that is integrated with MotionView and utilizes many aspects of HyperWorks to assist with the engineering of vehicle suspensions.
HyperCrash	CAE pre-processor tool developed to support the non-linear finite element solver, Altair RADIOSS.
AcuConsole	Powerful, yet easy to use GUI-Based Computational Fluid Dynamics (CFD) pre-processor for AcuSolve.

Aerospace 13.0 Release Notes

13.0 Release Notes

Ply Normal Correction to Match the Element Normal

CAD systems create different ply stacking directions for each ply stack (laminate). However, solvers can use only single element normal, hence if the ply stack direction and element normal are not matched, ply angles need to be corrected. This utility allows manual ply angle correction depending on the element normal.

Orientation Review

This functionality is used to display, review and modify frequently used coordinate system for analysis. Element, material and ply systems can be reviewed in a single location and modified. HyperView has similar functionality to make it a uniform user experience.

Beam Auto Property Calculation

This utility automatically calculates beam properties from a solid geometry model or FE model. You can define the FE beam elements and associate with the 3D CAD solid model to automatically retrieve the properties.

Ply Geometry Smoothing

Use this utility to convert FE ply geometry into geometry ply, and export it as STEP files using OS-smooth functions. These STEP files can be imported into CAD systems to recreate the ply. This is a very useful function for composite optimization using OptiStruct and for exporting the geometry for CAD design and manufacturing.

Rivet Connector

This utility connects two components (shell components) using a special aerospace rivet connection method using the Huth formula. This formula result depends on whether the components are composite or metallic.

Multiple Ply System

In previous versions of HyperMesh, the plies imported from CAD were allowed to use only one stacking system. This restriction has been removed, and now each ply can have its own system associated with it. The Ply Create dialog has been modified to accommodate this. The ply realization algorithm converts the ply angles based on these individual systems to a common system as the solvers currently accept single material systems for all the plies in each composite property.

Catia Composite Link (Simulyt interface)

Previous versions of HyperMesh were unable to import/export composite model with drape data from CATIA CPD modules. Only FiberSim composite import supported drape data from CATIA.

A new geometry import/export functionality from CATIA using Catia Composite Link is available. This new method is separate from the existing regular CATIA import with composite where drupe data is not imported.

Users need to use the CATIA composite (Simulyt) module, export an HDF5 file, and use that to import the file into HyperMesh using Catia – composite connection.

This is similar to the current FiberSim import /export method.

Airframe Modeling

This utility automatically creates Aerospace global finite element models (GFEM) from 3D CAD models. The resulting FE model consists of shell elements and 1D beam elements. The shell properties and BEAM properties are automatically derived from CAD models.

FE Absorb (ply absorb)

This utility converts zone-based composite models to ply-based models for OptiStruct. PCOMP and PCOMG models are automatically converted to PCOMPP (ply based model).

12.0.110 Release Notes

A new Aerospace user profile was added in the 12.0.110 release. This profile is aimed to help to capture the aerospace model building process and tools necessary for process automation.

Major Functions

- Element C1, C2, C3 direction: This to display 2D and 3D element local direction
- Free Body: This is to calculate Grid point force contribution to group of elements from the surrounding elements and break out modeling
- Map Forces and Mass: From an Excel or CVS file users can define nodal forces and mass and import into HyperMesh and automatically apply to existing model
- Matrix Browser: This new tool will help users to interrogate HyperMesh and HyperView data existing in the databases and export to Excel format for further calculations
- PCOMP from CSV: This is aimed to automate creation of large amount of PCOMP properties already defined in Excel or CSV file
- PCOMP to ply conversion: This utility converts zone-based models from Nastran to OptiStruct ply-based models
- Spatial Renumbering Elements/Nodes: This utility enables renumbering of nodes and elements in a predefined system directions for a regular structured mesh
- Resequence Nodes: This tool reorder the nodes in elements to achieve consistent C1 direction for a group of elements
- Shell to Solid conversion: This function converts shell composite elements into solid composite element for each ply layer
- Sewing tool: This connects a coarse Fe model to a Fine FE model using special RBE3 elements commonly used in Aerospace industry

CFD 13.0 Release Notes

13.0 Release Notes

New Boundary Layer Mesher

New boundary layer mesher **Smooth/Truncate BL** has been added in the CFD Tetramesh panel. There are two options, *simple* and *user controlled*, to define boundary layer parameters. In user controlled option, you can define two BL groups with different growth rate. The first group of BL is to capture the physics with slow growth rate. The second group of BL, which is optional, is useful for having a smooth transition from BL elements to tetramesh. Advanced settings are available to control BL propagations, proximity and quality of boundary layer elements.

Octree Based Tetra Mesher

New tetramesher option **octree based** has been added to the Tetramesh and CFD Tetramesh panel. It provides smooth element transition with good quality tetras. In general it is faster compared to the standard setting (advancing front).

Layered Tetramesh

In the Tetramesh panel, you can define the number of layers. The Tetramesher will ensure to have at least the defined number of layers, in case of close proximity or thin channels. It improves results in thin channels.

Adaptive Wrapper

The Adaptive Wrapper, accessed from the Mesh menu, is a useful utility to get a clean, water tight shell mesh out of 2D mesh containing several intersecting parts and small gaps which do not need to be modeled. A hole-gap patching functionality with a user defined threshold helps to automatically close openings. You can define element refinement based on proximity, feature/intersection lines and refinement boxes. Before wrapping process starts, the leak detection functionality can be used to check if the resulting wrapped surface mesh will create desired cavity or not with given settings. Optional automatic surface remeshing of the wrapping result ensures good surface mesh quality.

Boolean Operation

Boolean Operation located in the Mesh menu is useful to connect penetrating FE components. This tool works for triangular as well as quadrilateral elements. Several components can be connected simultaneously. Components defined as master entities will be preserved.

Enhanced AcuSolve Case Setup

The AcuSolve case setup has been substantially enhanced. The radiation model, mesh motion, reference frame, nodal initial condition can be defined in the HyperMesh Solver browser for the CFD user profile.

New AcuSolve Job Launcher

A new AcuSolve job launcher enables you to execute AcuSolve jobs directly from HyperMesh. You can access it via the CFD toolbar in HyperMesh. It integrates the mesh export and the AcuSolve job run. The submitted job can also be controlled from HyperMesh.

Smooth AcuSolve – HyperStudy Workflow

Automatic design exploration or optimizations can be performed using the HyperStudy Job Launcher located in the CFD toolbar. Once morphing shapes are defined and the AcuSolve case is setup in HyperMesh, the HyperStudy Job launcher transfers all the relevant data to HyperStudy. Afterwards, with a single click a design of experiment (DOE) or optimization study can be performed.

Surface Mesh Export in AcuConsole Native Format

The export interface for AcuConsole is now able to export a pure surface mesh in native *.arm format. This option is available via standard AcuSolve export as well as through the Launch AcuConsole icon in the CFD toolbar.

Resolved Issues

- Refinement box: Refinement box was creating intersections in surface mesh.
- BL meshing: Unnecessary faces and improper volume elements created after BL meshing.
- CFD tetramesh: Slow CFD tetrameshing compared to 11.0.130.
- Update of input shells: The input shells are not updated when the **BL only** option is used in the CFD Tetramesh panel for a mix of fluid (with BL) and solid volumes (w/o BL).
- HexCore Meshing: Improper volume mesh connectivity if quad elements are present in surface mesh.
- Mesh export for AcuSolve: Creates two layers of interface by default even if checkbox is turned off. Incorrect AcuSolve format mesh export, if model contains multiple volumes.

12.0.110 Release Notes

Boundary Layer Generation for Baffles and Sharp Edges

New option in the CFD tetramesh panel to control the boundary layer (BL) growth on the free edge of a baffle or for sharp edges pointing into the volume. For the free edge of a baffle, two normals can be used to grow the BL, resulting in a continuous BL around the free edge. For sharp edges a threshold can be defined, to decide whether two normals or one normal is used at the sharp edge to grow the BL.

Boundary Mesh Check for Tetrameshing

The **Check 2D Mesh** utility located on the **Tetramesh** and **CFD Tetramesh** panel can be used to validate the input surface mesh for volume meshing. Currently checks for intersection and dihedral angle (sharp edges) can be performed.

2D BL Meshing

Multiple options for default boundary selection in the **2D BL Meshing** utility.

Export Performance of AcuSolve Mesh Format

The performance for exporting in AcuSolve format has been significantly improved, up to three times faster. It is a direct export now, meaning no intermediate Nastran file is generated.

Create Exterior Shells During AcuSolve Export

If no boundary shells are generated to define the boundary regions, default exterior shells can be generated during the mesh export in AcuSolve format. This is an optional behavior and can be activated on the export tab for the AcuSolve export. Those elements will also be placed in a new component in the HM model.

Interface Shell Generation During AcuSolve Export

New option in the AcuSolve export tab to generate always two layers of interface shells between two adjacent volume zones.

Surface Mesh Export for AcuSolve

The export interface for AcuSolve is now capable to export a surface mesh in native *.arm format.

AcuSolve Exported File Format

Variable text field length for IDs to reduce memory consumption of *.crd, *.cnn and *.ebc files. Double precision for *.crd file.

Constrain for Boundary Layer

A new constrain to preserve the boundary layer thickness and layer distribution during morphing has been implemented. This ensures an unchanged y^+ (1st layer thickness) while morphing. The performance of this constrain has been optimized to deal with thick boundary layers, allowing quick design changes.

Large Model Morphing

CFD models are often characterized by a large volume element count. The new option "skin only" offers a two step approach for morphing those types of models in an efficient manner. In a first step, the user morphs only the boundaries of the flow domain interactively and the volume elements stay unchanged. When a new design has been found, the volume elements will be morphed in a second step in batch mode.

Morphing Performance

The morphing performance has been further improved by streamlining the existing code and by leveraging multi-threading technology. Those enhancements will be beneficial for large CFD models.

Enhanced AcuSolve Case Setup

The AcuSolve case setup has been substantially enhanced, e.g. thermal aspect, transient, turbulence modeling.

Fluent Mesh Import Containing Polyhedrals

Loading a Fluent mesh file (*.cas/*.msh) which contains polyhedral elements, will import all the standard elements (e.g. hexa, tetra) whereas the polyhedral elements will be ignored. Optional faces can be generated on the ignored polyhedral elements during import, so that tetra elements can be generated manually in this void region.

Duplicate Check for Export

A duplicate check for Nastran(Fluent), Nastran(PowerFlow) and Nastran (StarCD) export has been implemented.

Resolved Issues

- Refinement box: T-connections and feature edges included in a refinement box for tetra meshing will be remeshed according to the user defined element size for this refinement box.
 - CFD tetrameshing for baffles: Baffle shell elements are not duplicated when selected as "with BL (float)"
 - Renumbering during export: The node and element IDs are not renumbered during the AcuSolve mesh export.
 - Import/export issues: Improved the SC/Tetra, Fluent and OpenFoam import and export.
 - CFD tetramesh: Further improved the mesh quality and robustness for CFD tetra meshing as well as several bug fixes.
 - Update of input shells: The input shells are updated when BL only option is used in the CFD tetramesh panel for a mix of fluid (with BL) and solid volumes (w/o BL).
 - Separate component for solid mesh: In the CFD tetra mesh panel, when using "simple select" and uncheck "comp per volume", the solid mesh is placed in a separate component.
-

HyperView New Features

AcuSolve result reader: A result reader to import native AcuSolve results into HyperView has been implemented. Results of steady state, transient and moving/deforming mesh analyses are supported.

NVH 13.0 Release Notes

Introduction

To ensure successful adoption by customers, starting from 12.0, some of the high productivity NVH features will be packaged with additional services and made available through the NVH Solution of the Packaged Solution Offerings (PSO), and will not appear in the standard HyperWorks without a PSO license feature. See [13.0 NVH Packaged Solution Offering](#) for 13.0 updates specific to this package.

13.0 Release Notes

Model Correlation Utility

- Model geometry transformation
- Node pairing with extensive filtering capabilities based on sub-model, element configuration and distance tolerance
- Modal animation review and mode shape difference contour
- Modal Assurance Criterion (MAC) calculation and display supporting real modes, complex modes, and ODS results
- Partial MAC visualization to identify sub-models responsible for correlation issues
- Coordinate Modal Assurance Criterion (CoMAC) calculation and display
- MAC sensitivity studies by excluding nodes or sub-models based on CoMAC or PMAC values

Transfer Path Analysis Enhancements

- Summation of transfer paths using the Root Sum of Square (RSS) method for response calculation

13.0 - NVH Packaged Solution Offering

Modal Model Review and Parameter Modifications

- Support modal models of both CMS SE and Modal FE types
- Support frequency and damping modifications

Analysis Manager Enhancements

Enhanced solution types in Analysis Manager for generation of solver decks:

- Frequency Response – new functionality to generate solver deck for user defined loadcase templates supporting the automated generation of diagnostic requests
- Normal Modes – generate normal modes solver deck
- Full deck TPL – generate solver deck for user defined loadcase templates
- Solver deck
- Process Manager Generated – generate solver deck for process manager generated loadcase templates

Assembly and ID Management Enhancements

- Support cross reference management between include files that are not self contained
- Support tagpoint and connection FE entity ID lock
- Support assembly level ID reservation and module level ID subranges
- Skipping ID renumbering of all entities in a module or only for material and properties

Auto Generation of Loadcase Templates from a Loadcase PM Run

- Auto generation of loadcase templates from a normal modes loadcase PM run
- Auto generation of loadcase templates from a unit FRF loadcase PM run

Network View Coloring Capabilities

- Enhancement in Network Diagram with coloring capabilities to assist user in quick diagnostic related to missing representations for a module, connection types and realization

Mass Extraction Utility

- Extract module mass and moment of inertia information in prepare module mode
- Automated creation of rigid body representation using the mass and inertia information

Module Representation File Path Replacement

- Automatic mapping of modified and/or missing representation file paths and folders

Panel Set Manager

- New NVH custom entity to create Panel sets in prepare module

Integrated Diagnostic Enhancements

- Support for Energy Distribution and One Step TPA output as diagnostic results

12.0.115 Release Notes

Resolved Issues

- Resolved issue where the coarse display creation functionality in the batch prepare module feature always renumbered entities of the original model
- Resolved issue where switching from one module to another frequently within the edit module feature caused HyperMesh to freeze up

12.0.112 Release Notes

Resolved Issues

- Issues related to assembly XML file import using 'Child' and 'Replace' options
- Issue related to multiple AutoTPA requests in the General Frequency Response Process Manager
- Issues related to AutoTPA and Traditional TPA elements set definition in the General Frequency Response Process Manager
- Issue related to incorrect reference to module state MPC in the General Frequency Response Process Manager
- Issues related to multiple AutoTPA results (control volumes) support in the Transfer Path Analysis Utility

12.0.110 Release Notes

CDS SE Creation Process Manager

A new frequency response process manager for guided creation of CDS SE

Access to Job Manager in NVH Utilities

Added icon to invoke Job Manager and quick access to analysis results

Support for Units in NVH Utilities

Provide capability to show units information in axis labels

Enhanced Random Frequency Response Process Manager

- Support for PSDF flag for PSD output requests
 - Support for grid and element set based PSD requests
-

Enhanced General Frequency Response Process Manager

- Support for data in universal format for loadcase creation
 - Support for modal participation response type selection
 - Support for rigid element set for AutoTPA diagnostic output
 - Support for two step TPA
-

Enhanced Transfer Path Analysis Utility

- Support for matching file
- Support for reversing TF and force direction
- Support for test based transfer function and force data in universal format

Enhanced Integrated Diagnostics Utility

Support for test based response data in universal format

12.0.110 - NVH Packaged Solution Offering

HyperWorks 12.0 extends the offer for manufacturing and engineering solutions, with new functionalities and an innovative delivery model to better capture knowledge for vertical processes.

HyperWorks engineering solutions are now offered as packaged solution offerings (PSO), specialized packages of software and services that support the implementation and utilization of Altair's solutions and technologies, tailored to meet customers' specific requirements.

To ensure successful adoption by customers, starting from 12.0, some of the high productivity NVH features will be packaged with additional services through the NVH Solution of the Packaged Solution Offerings (PSO). Please contact an Altair representative to learn more about the available the Packaged Solution Offering and how to use it with your current HyperWorks licenses

Altair PSO for automotive NVH (noise, vibration and harshness) simulation includes:

Software

1. HyperMesh - NVH user profile under 'Engineering Solutions'
 - a. Assembly browser
 - b. Network view
 - c. Analysis manager
 - d. Job manager
 - e. Loadcase setup process managers in NVH user profile mode
2. HyperView/HyperGraph - Integrated Diagnostics utility

Training

1. Quick start video
2. Instructor-led class
3. Tailored course
4. On-site expert help

Customization

1. Generic
2. Special request

Project Engagement

1. Engage Altair ProductDesign NVH experts to execute a customer project

Automated Module Preparation

Provides capabilities for ID management and plotel display model creation

Automated Generation of Model Checkout Runs

A new dialog to generate jobs for model checkout runs of multiple modules

Automated CMS SE Generation

A new dialog to generate jobs for creating CMS SE of multiple modules

Automated CDS SE Generation

A new dialog to generate jobs for creating CDS SE of multiple modules

User Message Window

- Provide user oriented messages related to execution of user commands
 - Configurable message output
-

Macros Dialog

Provide capabilities to access macros packaged with the NVH solution and manage macros added by the user

CDS SE Creation Process Manager

A new frequency response process manager for guided creation of CDS SE

Composite Response Creation Tab

Provide capability to create new response points based on existing tagpoints

New Modeling Entities

Provides capabilities to define standard Response, Frequencies, Tire Patch Points for automated loadcase setup

Enhanced Module and Connection Browser Tabs

- Provide capability to open multiple instances of the Module and Connection Browser Tabs
 - Tabs can be floating as well as docked
-

Enhanced Tagpoint Related Functionalities

- Improved usability of the tagpoint tab
 - Enhanced tagpoint conflict resolution during XML import
 - Tagpoint creation using standard names from a csv file
-

Enhanced Analysis Manager

- Provide capability to override representation file path required on a remote server

- Export all bulk data to the master include file
 - Mapping files related to Module ID, connection, and tagpoint generated during job creation are moved to the output job folder
-

Enhanced LCS Manager and LCS Creation GUI

- Improved usability of the LCS manager
 - Provide capability to create new LCS entities based on data from existing LCS
-

Enhanced Connection Mapping Tool

- Export connection property to isolator file
 - Support for PBUSHT properties for import and export
-

Enhanced Lumped Parameter (LP) Representation ID Handling

LP representations are organized into the ID range assigned to the module, instead of that of the root module

Enhanced ID Manager

Capability to import and export module hierarchy and representation include file information in the module ID CSV file

Enhanced Network View

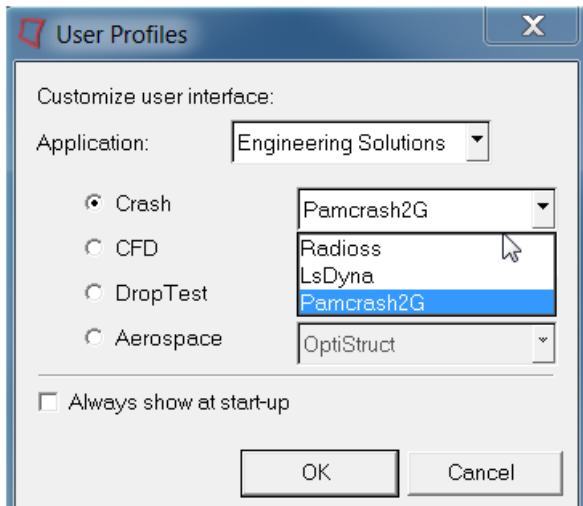
- Capability to automatically expand connections between two modules
- Enhanced entity information in message bar with mouse hover

Crash 13.0 Release Notes

13.0 Release Notes

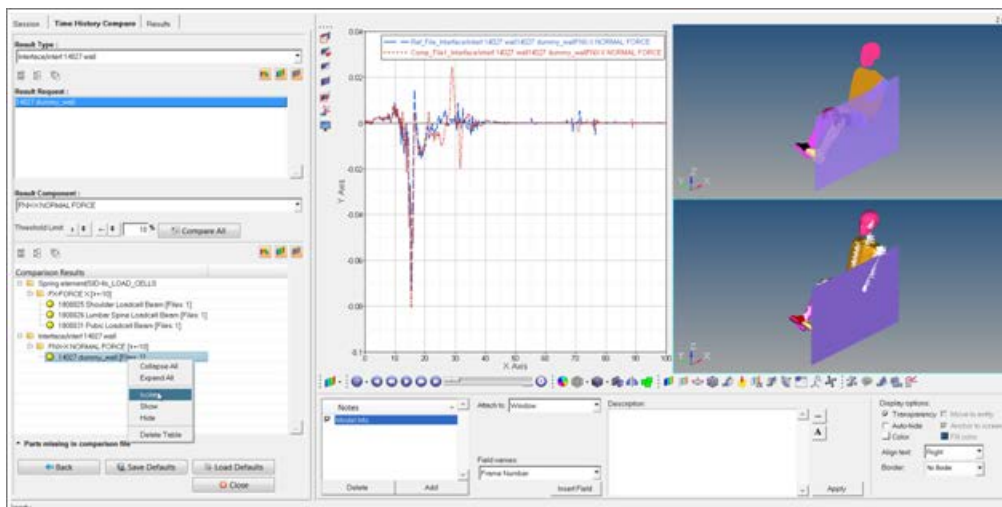
Engineering Solutions in Desktop

Earlier, Engineering Solutions profile for Crash RADIOSS and LS-Dyna was available in the stand alone HyperMesh profile. Now we have these profiles available in Desktop mode. Also in this release we have introduced for Pam-Crash. In Engineering Solutions profile, the drop down menu bars of HyperMesh have been customized to make it easy for crash users.



Compare Time History

This tool is available in HyperView with Vehicle Safety Tool preferences. It allows the comparison of time history data for RADIOSS and LS-Dyna as well as visualization in HyperMesh and HyperView. Time History output of RADIOSS (T01) and LS-Dyna (binout and other ascii files) are supported. Several results file can be compared against a reference file using a criteria. Currently the percentage difference between max of curves and at every data point of a curve is supported. The results of the comparison is shown in HyperGraph, HyperMesh and HyperView.



Pedestrian Protection – Upper and Lower Leg

In 13.0, added support for upper and lower leg impactor

- Vehicle Marking is more robust and flexible with support for wiper components. (only for HW13.0)
- User interface is enhanced to be more user friendly interface.
- Vehicle Positioning task modified for multiple transformations.
- Master Decks size reduced significantly with new organization.
- Contact and transformation cards includes are created separately.
- Engine file preview added; it can be modified using the built-in text editor.

Move to Include

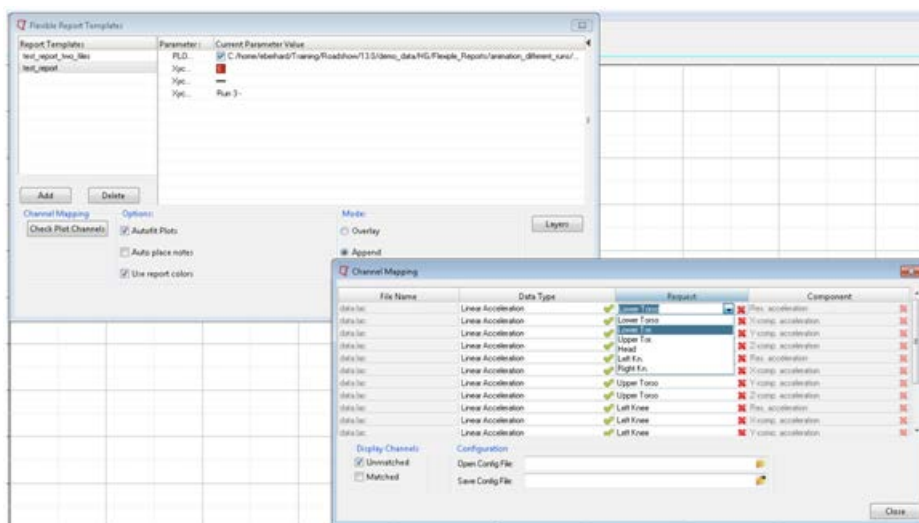
In earlier versions, in the Model browser, the **Organize Include** context menu was available when one or more components were selected in the browser. Now this feature has been renamed to **Move to Include**. It is also available in the Include view of the Model browser.

Organize Include

In the Include view of the Model browser, at the include file level, the **Organize Include** option is available in the context menu. This is available only for LS-Dyna. It gives users an option to make the include self-contained by retaining only the entities related to the components in that include. Please note, for large models, it can take several minutes. Performance will be addressed in a future release.

Flexible Report Templates

The new easy-to-use report Flexible Report panel replaces the Report panel from earlier HyperWorks Desktop versions. It makes reports more flexible by giving the user advanced options to handle channels during the import. All channels referring to files can be skipped; their types, requests and components can be checked and reconnected in case there is a mismatch. The reconnected channels can be saved and reused using configuration files. More user-friendly color and style settings are possible for parameters created with the Parameter browser.



Marker Tracking

The Marker Tracking feature in MediaView allows the user to track any kind of markers or objects in MediaView as the video is animated. This provides numerical data which can be displayed and animated in HyperGraph. Manipulation of the tracking data by applying units or other math operations allow the comparison of tests and simulations. Markers can be added via mouse or API. Graphical options like show/hide tracking lines and marker IDs, or show tracking markers and traces makes it easy for the user to analyze the testing video.



CAD and Solver Interfaces 13.0 Release Notes

Use the links below to view the HyperWorks 13.0 release notes for the CAD and solver interfaces.

[OptiStruct](#)

[RADIOSS](#)

[Abaqus](#)

[ANSYS](#)

[CAD Interfaces](#)

[LS-DYNA](#)

[Nastran](#)

[PAM-CRASH 2G](#)

[Samcef](#)

Abaqus Interface 13.0 Release Notes

13.0 Release Notes

Smoothed Particle Hydrodynamic (SPH)

- SPH is supported in the Abaqus/Explicit user profile. Typical applications are for modeling airbags, fuel tank slosh, bird strikes, and explosion analysis. It is currently implemented in the SPH panel.
- Finite element mesh can be converted into SPH either at the beginning of an analysis or after the deformation becomes significant. The *section control card is updated to support this feature.

Entity Editor

All the HyperMesh entities in the Abaqus interface can be created and modified from the Entity Editor.

Nodal Thickness Visualization

- **2D Detailed Element Representation** setting and **By Thickness** option has been updated to visualize nodal thickness assigned to elements.
- 1st and 2nd order elements of trias and quad are supported in this implementation.

Abaqus Model Checker

New warning has been added to identify empty contact pairs or ties. This check identifies if either master or slave surface is not selected for the contact pair.

New Keywords

- *MODAL DAMPING
- *FAIL STRESS/*FAIL STRAIN
- *TIME POINTS
- *INITIAL CONDITION

Resolved Issues

- Preview empty option in the Delete panel was not working for the group entity. All the entity's in group which as a component, set or surface to be selected will be captured with the **Preview Empty** option. System that are not used in the model can be identified using the **Preview Unused** option in the Delete panel, the issues with this option are fixed. Currently the Preview Empty option for groups and Preview Unused option works as expected.
- For *Section Control keyword a new option **Drill stiffness = off** is added.
- For *Energy Output Keyword a new option **PER SECTION** is added. The same is also added in the Step Manager.
- For *CONNECTOR BEHAVIOR keyword, the option EXTRAPOLATION=LINEAR was lost during export and import. No warning message was issued. This option is fixed for *Connector Behavior.

- For *Connector Elasticity keyword a new option **Independent Component = Position** is added.
- During import and export the *DAMAGE INITIATION keyword under *COHESIVE BEHAVIOR was getting assigned to *Material. The *DAMAGE INITIATION is added under the unsupported keyword for * COHESIVE BEHAVIOR.
- The Node and Nodeset option in *Clearance is allowed to select a node instead of typing the coordinate value aX(1)...bZ(1) via node selection.
- Abaqus\Explicit Step Manager will support *Bulk Viscosity keyword.
- For *Section Control keyword a new option **Drill stiffness = off** is added.
- For *CONNECTOR MOTION keyword the option, TYPE=DISPLACEMENT was lost during import and export. No warning message was issued. This issue is fixed for * CONNECTOR MOTION.
- Surface based on element can have blank line next to *Surface keyword. Import and export of *SURFACE with blank is supported.
- For *DSLOAD, *DLOAD & *CLOAD the Load Case parameter for Steady State Dynamic is removed. From Abaqus 6.12-1 onwards this parameter is not supported.
- HvTrans was not reading specific contact results for specific ODB. The HvTrans reader is updated to import all Contact Results.
- For specific ODB files, the CPRESS and CSHEAR result were not imported in HyperView. The HyperView Abaqus ODB reader is updated to import COPEN and CSHEAR results.
- *INITIAL CONDITIONS option will support both TYPE = REF COORDINATE and TYPE = REF NODE COORDINATE.
- For *CONTACT PAIR keyword a new option **Drill stiffness = off** is added.
- During import and export the *CONTACT PAIR loses the MIN DISTANCE = ON option. This issue is fixed for Abaqus/Explicit profile.

12.0.115 Release Notes

New Features and Enhancements

- *TIME POINTS keyword is supported as a group entity. This can also be referred in the *OUTPUT keyword.

Resolved Issues

- If an element ID is greater than 9, then *surface was not exporting proper data into input deck. The element ID limit is increased to 16.
- The name provided for *Distribution Table is retained during import and export.
- Apart from Length/Ratio/Drape, any unsupported *Distribution entity is moved to an unsupported card.

12.0.112 Release Notes

Abaqus to OptiStruct Conversion

- CONN3D elements to CBUSH/PBUSH.
- Distributed Coupling surfaces to RBE3.
- Offset definitions for composites using SNEG/SPOS keywords.

- Orthotropic material to MAT8.

Newly Supported Keywords

- *Fail stress keyword is supported for Abaqus Material.

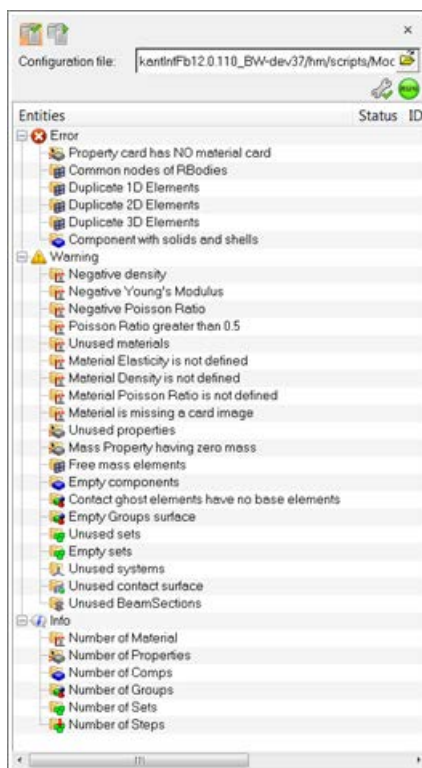
Resolved Issues

- Abaqus to OptiStruct Conversion
 - Conversion of Abaqus *MPC PIN to OptiStruct RBE2 without DOF values issue has been corrected.
- Abaqus Interface
 - Defect while exporting an input deck with a MPC Beam converted into a Kinematic or Distribution coupling.

12.0.110 Release Notes

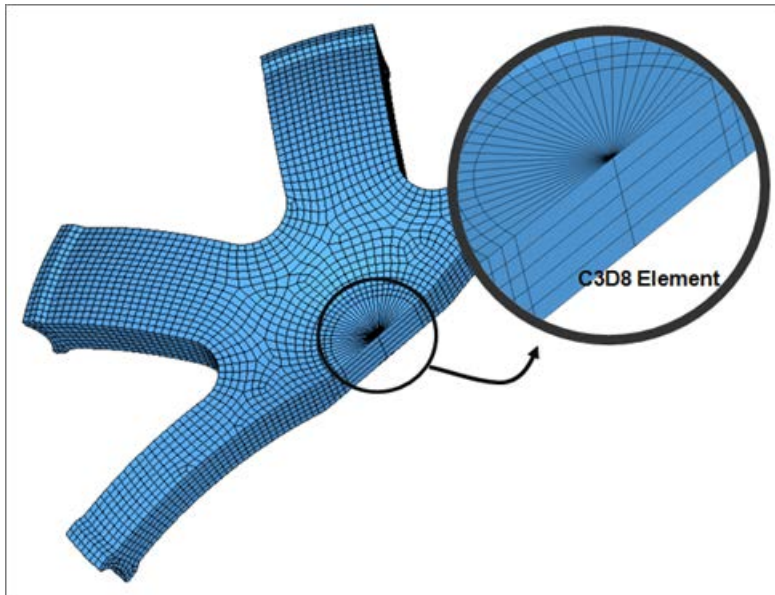
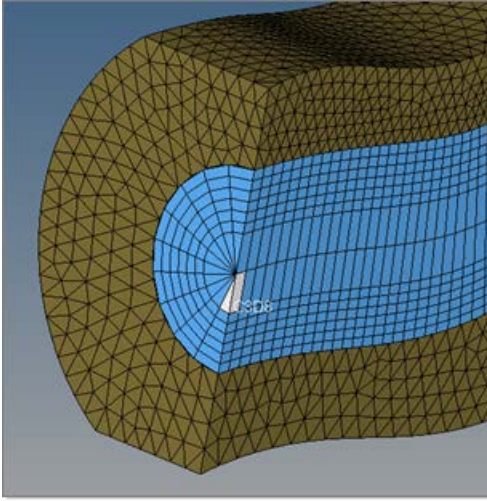
Abaqus Model Checker

- Tool to identify modeling errors/warning. This is added as complimentary for the current element quality check.
- Currently implemented 6 errors, 19 warnings and 6 model information checks. This tool allows you to manually or automatically fix errors or warnings.
- Completely customizable environment. User-specific checks can be added easily.



Degenerated Hex Element

- Degenerated brick elements are either Hex8 or Hex20 elements, collapsed into a wedge element (Penta6 or Tet15).
- Standard node order for a 1st order, Hex8 (C3D8) element will be 1, 2, 3, 4, 5, 6, 7, 8. In cases of Degenerated Brick elements, the node order will be 1, 2, 3, 3, 5, 6, 7, 7, with nodes 4 and 4 being repeated during export.



FiberSim Support

- A Fibersim data reader enables users to read results of a draping simulation
- Fibersim data can be used in a new process to automatically transfer geometric composites information to finite elements for huge time savings in modeling time
- The ply information from FiberSim can be realized and assigned to the mesh created in HyperMesh
- More options are added to SHELLSECTION_COMPOSITE property to support the data from FiberSim
- The table entity will be used to assign the drape data to the element

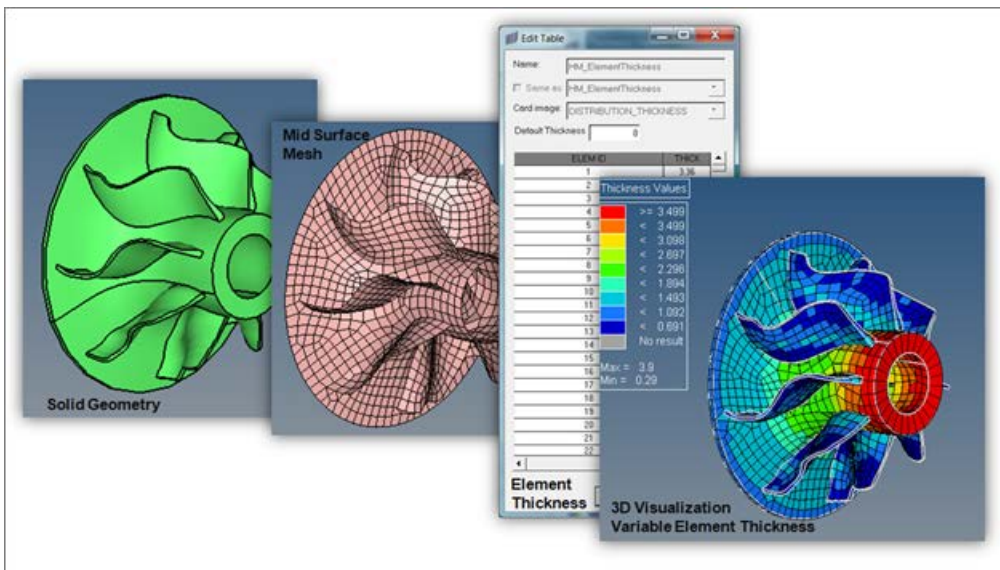
Table Entity

A new entity is added to support *DISTRIBUTION support. The **table** entity can be created from the Solver Browser. The table is assigned to elements through property. Element will fetch data from the table that is assigned to the property. Three card images are added to the table entity.

- DISTRIBUTION_Thickness
 - Define the element thickness
- DISTRIBUTION_Offset
 - Define the element offset
- DISTRIBUTION_Drape
 - Define thickness and ply angle



The element value can be edit from the Edit Table dialog. 3D visualization the assigned property is enabled. 3D visualization is added to entity which also user to visualizes and check the element thickness.



Abaqus ID Project

- Each entity in HM as an ID, in case of Abaqus Interface the solver deck exported from HM will not retain those ID's
- OEMs are working with strict ID ranges, E.g. Engine-system always has Part ID 2000000-2100000 across all carlines. So an engineer always knows that he has to isolate PIDs 2000000-2100000 if he wants to examine the Engine-system. This is essential in pre- and post-processing.

- Example for a surface entity with ID's in solver deck (HM group):
 - *SURFACE, NAME = <Prefix><ID>; <name_of_surface>
 - Example for a surface entity without ID's in solver deck (HM group):
 - *SURFACE, NAME = <name_of_surface>
-

Contact Manager

- Added sorting functionality
 - Review contacts based on the color selected for that surface
-

Bars Panel

- The offset and section axis defined in the Beam section property will be used for beam creation.
 - Either a property should be assigned to the component or during the beam creation the property must be selected.
-

Abaqus ODB Reader Support- HyperView

- Abaqus 6.12-1 reader has been added to the already existing 6.11-1, 6.10-EF and 6.10-1 readers.
 - The HyperView Results Browser will reflect the assembly structure when reading an Abaqus .odb file with *Part and *Instance
-

General Enhancements

- Support for create, import and export membrane element M3D4 elements in Abaqus Explicit-Template
 - Traction type is added to the material definition
 - Elements option is linked to the Table. For each shell element thickness is calculated
-

Resolved Issues

- The Preview Unused system option in the Delete panel is fixed to capture systems used in *transform option
- The Tracking thickness option in *surface interaction can be exported/imported
- CONTACT THICKNESS REDUCTION=NOPERIMSELF option in * CONTACT CONTROLS ASSIGNMENT can be exported/imported
- The constraint name of the coupling will be preserved during import and export of Abaqus input decks
- Changing the order of a beam from linear to quadratic and again back linear, will no more change the element type to unsupported.

ANSYS Interface 13.0 Release Notes

13.0 Release Notes - Pre-processing

Supported Versions

HyperMesh 13.0 supports version ANSYS 15.0 and earlier.

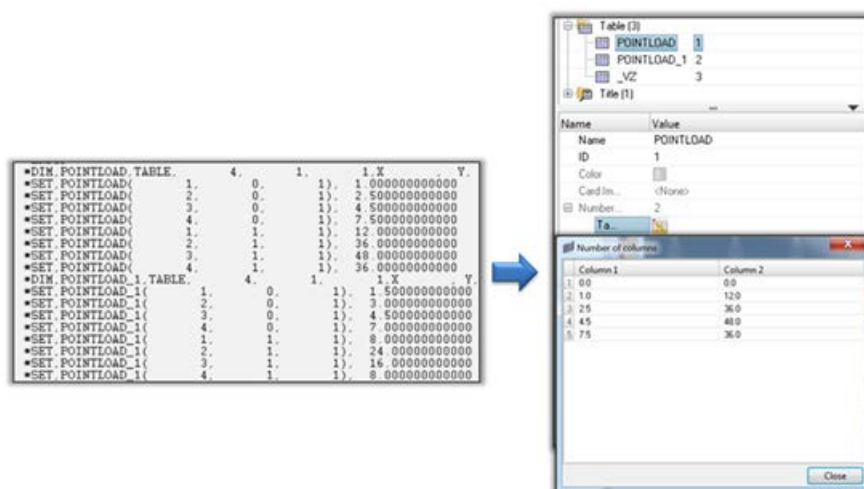
ANSYS TABLE and ARRAY Support

ANSYS solver has multiple values parameters called Arrays. Data like loads, time, location, frequency, can be stored in these multidimensional arrays and then used in the models for applying loads.

Supported Array types:

- TABLE: Typically used for load application, where index values can be real numbers.
- ARRAY: Index values are integers. Parameters can be real or integers.
- CHAR: This is a character array consisting alphanumeric values not exceeding to 8 characters.
- STRING: Array with character Strings.

In HyperWorks 13.0, TABLE, ARRAY, CHAR and STRING data defined using *DIM and *PREAD keywords are supported. HyperMesh can import and export the array data. All of the array types are supported as tables in HyperMesh. You can review and edit the arrays imported by clicking **Model Browser > Tables > Entity Editor**. HyperMesh supports one dimension and two dimensions arrays. Loads that are referred to TABLES are preserved as unsupported data. When exporting from HyperMesh, such load information will also be exported.



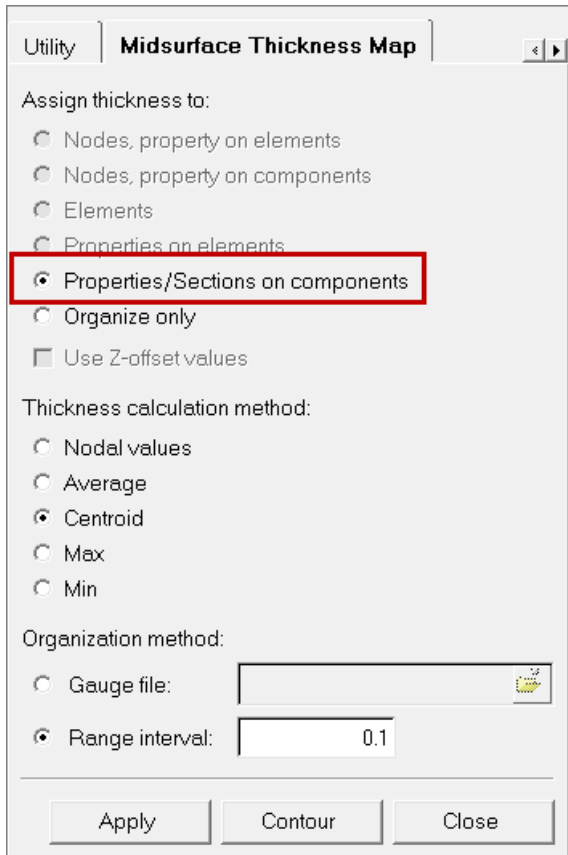
Midsurface Thickness Macro

The ANSYS interface in HyperMesh includes a utility to extract model thicknesses from a geometry model and apply the thickness values to FE shell elements. This tool can be accessed from **Utility menu > Geom/Mesh > Midsurf Thickness**.

In previous releases this tool could be used only for those shell elements in which thickness is applied using property (REAL set). For example, SHELL63, SHELL93, SHELL43.

In HyperMesh 13.0, this utility supports new versions of ANSYS shell elements such as SHELL181, SHELL281 and SHELL131. Shell thicknesses for these elements are defined by shell sections (SECDATA). Using this utility creates shell sections and assigns thicknesses to those sections. These shell sections are attached to the components of respective shell elements.

However the tool also works for ANSYS older shell elements where shell thickness is defined by properties.



Entity Editor

The Entity Editor is customized to ANSYS profile. The Entity Editor can be accessed from the Model and Solver browser. ANSYS solver information such as element types, key options, properties (REAL SETS), materials, section data, control cards, and entity sets can be added or edited from the Entity Editor. Some important highlights of the Entity Editor are:

- Components, element types, properties (REAL sets), materials and section cards can be created or edited and then attached to the components in one Entity Editor dialog. There is no need to use different dialogs for each of these entities.
- Tool tip help is provided for element key options definitions and properties to help you choose the right option. This avoids you from having to refer to solver online help. Tool tip help can be seen by moving the cursor on the key options in element type or property options in property card.
- You don't have to remember the KEYOPTION values to input under every KeyOpt you select. A drop-down menu is provided for each KEYOPTS to select the appropriate option for every element that HyperMesh supports.
- Checks are provided in the Entity Editor to ensure the correctness of the component with respect to element types and properties by displaying errors in message bar. Appropriate default values, suggestions are provided where ever applicable.
- Temperature dependent material data input has been simplified. The need for two tables, one for temperatures and another for the corresponding material values, are no longer needed.

Name	Value
Name	TYPE3-REAL22-MAT1
ID	21
Color	[Red]
Card Image	HM_COMP
Type	ET_3 (3)
Property	PROP_22 (22)
Material	MAT1 (1)

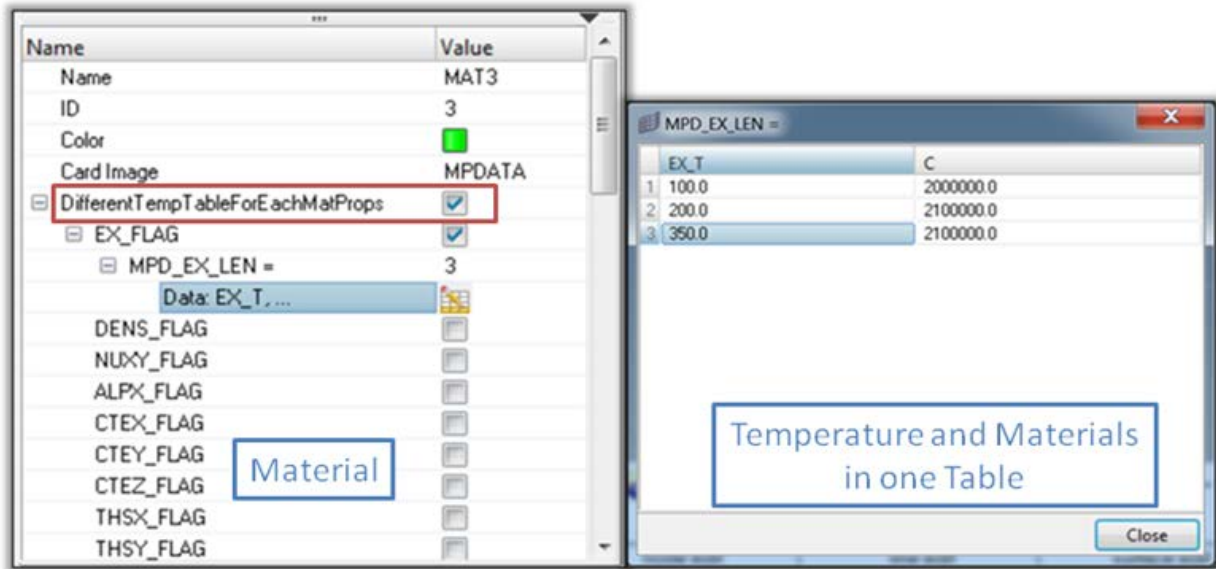
Component

Name	Value
Color	[Green]
Card Image	SHELL181p
Real Constants	
TKJ	3.0
TKJ	0.0
TKK	0.0
TKL	0.0
THETA	0.0
ADMSUA	0.0
E11	0.0
E22	0.0
E12	0.0
DSF	0.0
MHGF	0.0
BHGF	0.0

Property

Name	Value
Name	ET_1
ID	1
Color	[Red]
Element Type	SHELL181
KeyOpt1	<input checked="" type="checkbox"/>
value	0-Bending and membrane stiffness
KeyOpt3	0-Bending and membrane stiffness
KeyOpt8	1-Membrane stiffness only
KeyOpt9	2-Stress/strain evaluation only
KeyOpt10	<input type="checkbox"/>

Sensor (Et Type)



Training Program on the ANSYS Solver Interface with HyperWorks

We have introduced new training material that primarily focuses on the ANSYS solver interface. Training material covers the complete model preparation for solving in ANSYS and then post processing the output files using HyperView.

This training program is useful for those who know HyperWorks tools but are not familiar with the ANSYS solver; for those who know ANSYS but are new to HyperMesh and HyperView; and those who want to use HyperView for post-processing the ANSYS result files.

Salient features of the training program include:

- Pre-processing, post-processing and advanced modeling tools, including Contact Manager, composite modeling, and HyperBeam.
- Ample number of exercises and demonstrations in each chapter.
- Customizable to individual interests.
- Discusses new features and processes like the Entity Editor as well as ply and laminate modeling.
- Highlights the differences in naming FE entities between HyperMesh and ANSYS to benefit users who are familiar with ANSYS.

Interested users can contact Altair support for more details and scheduling the training.

Resolved Issues

- Issue with material orientation for shell elements in graphics where if a local system is assigned to shell elements then the graphics shows incorrect orientation is now fixed.
- Issue with material export where some material data is exported twice has been fixed.
- Issue with defining surface to surface contacts on pyramid element faces using ANSYS component manager has been fixed.
- Performance issue with export of models with large number of components or with large number of sections is now fixed.
- ANSYS CDB files with node system angles in UNBLOCKED (old format) are not imported in HyperMesh with correct angle values is fixed now.
- Pretension Bolt utility was not working with 1-D bar elements created with direction node option is fixed.
- 3D visualization of Shell elements with thickness defined by shell sections now considers the offset values defined by SECOFFSET.

13.0 Release Notes - Post-processing

Supported Versions

HyperView 13.0 supports version ANSYS 15.0 and earlier.

Resolved Issues

- Issue with composites shells with section definition where results for layers are swapped between layers is fixed now.
- Issue with stress results for second order tetra elements and issue with frequency values for complex modal analysis where all frequencies are listed as zero for the result files from ANSYS14.5 version are fixed.

12.0.115 Release Notes

Resolved Issues

- Issue with pretension bolt utility in HyperMesh where the tool was not working for BEAM elements created with direction node option is fixed.
- Issue with loads type: constraint where extra load constraint was exported is now fixed.
- Issue with composites shells with section definition where results for layers are swapped between layers is fixed.
- Issue with tensor results missing for solid elements in the models that have composite shell elements is fixed.

12.0.112 Release Notes

Resolved Issues

- Issue with Constraint Equations where equations with thermal labels (DOF) were not handled by HyperMesh in earlier versions has been fixed now.
- Issue with export of load steps where loads applied on same node in same direction were not combined is now fixed.
- Issue with derived load cases exporting to H3d file where results for shell elements were missing is now fixed.

12.0.110 Release Notes

ANSYS v14.5 Version Support in HyperMesh and HyperView

ANSYS solver input deck format and result file format have been changed in its latest version: ANSYSv 14.5. HyperMesh 12.0-110 supports this new format. Input decks from ANSYS 14.5 can be read in HyperMesh now.

ANSYS changed its result file format too. These result files can now be post processed in HyperView 12.0-110.

Ply and Laminate Support for ANSYS Models in HyperMesh

In HyperMesh 12.0-110, ply and laminate entities are supported for ANSYS models. User can create composite models based on ply and laminate definitions. Also ply and laminate data can be imported from FiberSim files. On realization of the ply and laminate data, ply based model will be converted to zone based composite model. ANSYS composite properties and shell sections will be created. Also by defining composites by ply and laminates, visualization of plies is also supported.

Pretension Bolt Utility

In HyperMesh for ANSYS profile a new utility –“Pretension Bolt” is now available under the Tools menu. This tool can be used to create pretension loads in bolts, and can be used for both 1D bolts and 3D bolts simulations. You can create ANSYS Pretension elements –PRETS179, Pretension sections and pretension loads. User can pick multiple bolts to load in single operation.

Other Enhancements

- New Element Support: Radiation elements SURF251 and SURF252 are now supported in HyperMesh.
 - Radiation load support: Radiation load- RDSF in surface load (SFE) card is now supported in HyperMesh. This load can be applied using the Pressure panel.
 - /TITLE card is now available under each load step. This can be used to name each load step. Load card OMEGA is also available for each load step.
 - New cards support: Radiation cards – RADOPT, STEF, HEMIOPT, SPCTEMP.
 - OptiStruct/Nastran to ANSYS conversion tool improvements – Conversion of MATT1, MAT1, conversion of CBUSH, CELAS, CGAP and conversion of PBUSH, PELAS, PGAP have been added in this release.
 - Post-processing of composite results – HyperView now post processes results for ‘all layers’ of composite elements defined by shell section option. Please note that this feature is not available for the result files created from ANSYS12 or earlier versions.
 - New results support: Failure criteria results based on stresses in damaged material for composites are now supported. Following failure criteria results are now available - Maximum of all failure criteria(MAX), fiber tensile failure (FT), fiber compressive failure (FC), matrix tensile failure (MT) and matrix compressive (MC).
 - ETABLE results support: Following elements are now added under ETABLE results - Fluid116, Link31, Link32, Link33, Link34, Mass71, contact170, Contact 173, Solid70, Solid226, Solid227, Solid87, Solid90, Solid185-187, Plane55, Plane13, , Plane223, Plane35, Plane77, Plane182, Plane183, Shell281, SURF151, SURF152, SURF251, SURF252
-

Resolved Issues

- Issue with Auto contact where in few cases contact surfaces were not recognized has been fixed.
- Issues with mass calculations of MASS21 element type have been fixed.
- Issue with more than one SECJOINT card where HyperMesh used to import only the first card has been fixed. Now HyperMesh can import more than one SECJOINT cards.
- Issue with beam elements where pin flag value is set to 3 even though no value has been assigned has been fixed.
- Issue with Load case label mismatch in HyperView in Linux platform has been fixed.

CAD Interfaces 13.0 Release Notes

13.0 Release Notes

New Features and Enhancements

- UG NX 8.5 (native and third-party readers)
- CATIA v5-6R2013 (v5 R23)
- Creo 2.0 M060
- SolidWorks 2013
- Support of JT export from HM of geometry and FE data

12.0.115 Release Notes

New Features and Enhancements

- Support for UG NX 9.0 (native reader)
 - UG NX 9.0 now requires the license server to be defined using the SPLM_LICENSE_SERVER environment variable, while older NX versions still require the UGS_LICENSE_SERVER environment variable.
- Users can now identify specific parts to include/exclude during JT import using the @FilterByJTOpenParts option in the jt_reader.ini file

12.0.110 Release Notes

General

- Parasolid Native CAD import is added
-

ACIS

- Version R21 is added
-

Catia

- Version R10 to R22 is enabled
-

DXF

- AutoCAD 12 is added

IGES

- 6.0; JAMA-IS is available
-

JT

- 9.5 support is available for import and export
-

Parasolid

- V20 support is added for non-native import
 - V25 support is available in native reader
-

ProE

- Version 13 and CREO 2 support is added
-

SolidWorks

- Version 2012 is supported
-

STEP

- AP203/AP214 is supported
-

UG

- Up to UG NX 8.0 is supported using non-native interface
 - Up to UG NX 8.5 is supported using native interface depending on UG-NX installation (MAC is not supported)
-

VDAFS

- V2 is supported

LS-DYNA Interface 13.0 Release Notes

13.0 Release Notes - HyperMesh

New User Profile

- Keywords supported until 12.0.115 available as LSDYNA 971 user profile.
- New user profile LSDYNA971_R6.1 is now available, supporting all new modifications based on solver version 971 Release 6.1.

Midsurface Meshing

- Thickness information from CAD data can be mapped to either the midsurface element in the current session or to an Include file referred as *INCLUDE_STAMPED_PART in the current session.
- Support for node based and element based thickness mapping.
- Support for contour and 3D visualization of thickness calculated from Midsurface Mesh tool.
- Support for offset and scaling capabilities for thickness calculated from the Midsurface Mesh tool

Entity Editor

- Support for all keywords mapped in the LS-DYNA user profiles except elements.
- Create options available for all keywords mapped to named HyperMesh entities, such as component, property, material, interface, rigidwall, airbag, load collectors, system collectors, vector collectors, sets, block, sensors, and output blocks.
- Edit option available for all keywords except keyword mapped to elements in HyperMesh.
- Reference entities in keywords
 - New entities that are supported in Entity Editor can be created on fly through a dialog from the current keyword opened in Entity Editor. The new created keyword is referred back in the current keyword.
 - Referenced entities in the keywords can be edited on fly through a dialog from the current keyword opened in Entity Editor.
 - New Entity Selection browser is introduced to select exiting entities in session with options to filter based on name, ID, card image, include file, and so on.
 - Entity Selection browser by default is auto filtered to show entities of valid solver keywords for selection to create error free solver keywords.
 - Show/hide/isolate/Xref functions are available for each referenced entity in the Entity Editor for quick graphics review.

Model Browser

- Solver specific show/hide/isolate/isolate only functions extended to Model browser like in Solver browser.

Support of *PARAMETER and *PARAMETER_EXPRESSION

- Import/export support for parameter reference using & , < > and Negate parameter reference.

- New Parameter Expression created upon import for parameter references using , <> and sign and exported.
- Create, edit and reference of *PARAMETER in other keywords using Entity Editor.
- *PARAMETER_EXPRESSION support limited to import/export.
- Create and edit options are not available for *PARAMETER_EXPRESSION
- *PARAMETER_EXPRESSION evaluation during import and in session if any parameter values are updated.
- Reference of *PARAMETER_EXPRESSION in other keywords using Entity Editor
- Support is limited to all FE entities except elements.
- Parameterization of entity IDs and attribute of type entity are not supported.
- Unsupported parameterized entities are replaced with corresponding values during import, to avoid the loss of data.
- The variables parameterized will be grayed out in the card editor so that it cannot be changed.
- In macros and browserw, if any parameterized variable is changed a warning message will appear in the message bar and the value will not be updated.
- Support for undefined parameter references and duplicate parameter names.

Export Options

- Repeat keyword titles - Export writes keyword title for every occurrence of the keyword

Import Options

- Generate element property ID's per include file
 - Generate ID's for solver keywords mapped to HyperMesh elements and properties based on the max ID of the elements and properties in the destination include file.
- Extend Element, Property Offset
 - Applies offset defined for elements (IDEOFF) and (IDROFF) properties in INCLUDE_TRANSFORM keyword to other keywords mapped to HM elements and properties. This includes elements : *CONSTRAINED_JOINT , *CONSTRAINED_NODE_SET , *CONSTRAINED_NODAL_RIGIDBODY, *CONSTRAINED_GENERALIZED, *CONSTRAINED_SHELL_TO_SOLID, *CONSTRAINED_SPOTWELD, *CONSTRAINED_RIVET, *CONSTRAINED_INTERPOLATION and properties : *INTEGRATION_SHELL, *INTEGRATION_BEAM, *CONSTRAINED_JOINT_STIFFNESS, *MAT_ADD_EROSION, *DAMP_RELATIVE.
- Read *INITIAL_STRESS_SHELL and *INITIAL_STRAIN_SHELL
 - Read these keyword into HM if it is turned on. When turned off this keyword is moved to the .hmx file and the .hmx file is attached as an Include file to the session with Skip option.
- Include Files – Skip, Merge
 - Supports **Skip** and **Merge** option for include files referred as *INCLUDE_TRANSFORM and *INCLUDE_STAMPED_PART.

Import Reader Enhancements

- Warning messages added for any unsupported fields or unsupported lines encountered while reading material, properties, and control cards from solver deck.
- Reader performance enhanced to handle large models with large number of sets and contact surface.
- Support for multiple reference of same include file through *INCLUDE and INCLUDE_TRANSFORM in any order/sequence in the input deck.
- Support for node set information in *Assembly keyword used in dummy tree definition.

- GPP encrypted data is now mapped to encryption entity.
-

Find Attached (Tied)

- Solver based find attached feature that also includes following keywords to establish attachments: *CONTACT_TIED, *CONSTRAINED_EXTRA_NODES, and *CONSTRAINED_RIGID_BODIES
-

New Checks in Model Checker

- Tied contact check added to review and correct untied nodes.
 - Reference of empty set, component, contact surface in *CONTACT keywords.
 - Reference of set of other types in *SET_NODE_ADD.
-

New Keywords Supported

- _MORTAR option in the *CONTACT keywords
 - _CONTRACTION_JOINT option in the *CONTACT keywords
 - _THERMAL_FRICTION option in the *CONTACT keywords
 - *CONTACT_2D_AUTOMATIC_TIED
 - DATABASE_BINARY_D3PART
 - DATABASE_EXTENT_D3PART
 - *MAT 130 - *MAT_SPECIAL_ORTHOTROPIC
 - *MAT_COHESIVE_MIXED_MODE_ELASTOPLASTIC_RATE
 - *STRESS_INITIALIZATION
 - *STRESS_INITIALIZATION_DISCRETE
 - *STRESS_INITIALIZATION_SEATBELT
-

Updated Supported Keywords

- *CONTROL_HOURLASS
 - *CONTROL_SPH
 - *MAT_LAMINATED_COMPOSITE_FABRICE
 - *MAT_NONLINEAR_ELASTIC_DISCRETE_BEAM
 - *MAT_SIMPLIFIED_RUBBER/FOAM_WITH_FAILURE
 - *MAT_FU_CHANG_FOAM
 - *SECTION_BEAM
 - *SECTION_SOLID_EFG
 - *SET_PART_ADD
-

Resolved Issues

- Curve ID parameterized in materials lost during import.
- Mismatch of ID's of *HOURLASS upon import with FEOVERWRITE.
- Wrong error messages for *CONSTRAINED_EXTRA_NODES when reading decks with *INCLUDE_TRANSFORM.

- Scale factor in *BOUNDARY_PRESCRIBED_MOTION_RIGID changes to 1.0 on importing HM12.0 file.
- Model Checker LS-Dyna: Xtrancode PID not defined error is not working.
- *DEFINE_COORDINATE_SYSTEM not exported after creation from System panel.
- Unable to import *CONSTRAINED_SHELL_TO_SOLID.
- Data lost during export for *CONSTRAINED_TIE-BREAK.
- *ELEMENT_BEAM not populated in Solver browser upon creation.
- "ALE_REFERENCE_SYSTEM_SWITCH" keyword created with large ID.
- Mass values in *ELEMENT_MASS resets to zero when 3D representation is turned on before export.
- Reading *ELEMENT_SHELL_THICKNESS and *SET_SOLID_TITLE keyword in free format.
- Box Trim: Constraints created at the cutting plane even the constraints option is turned off.
- Box Trim: Box Collector with block or HEX entity is not created after Box Trim process.
- Loss of *Load_Node_Point (Forces) during export in models with *INCLUDE_TRANSFORM.
- ID of the set attached to rigid link changes and moved to bottom of the list in Model/Solver browser when rigidlink element is updated.
- HM exports two node sets with same ID
- Additional lines not in manual are available for keyword *CONTACT_2D_AUTOMATIC_SURFACE_TO_SURFACE"

13.0 Release Notes - HyperView

New Features/Enhancements

- Support for SPH results in FEMZIP reader.
- Support of new data type Dtemperature/Dt in FEMZIP reader.

Resolved Issues

- Names of components in Component browser is not as per solver deck upon model and result file reading.
- FEMZIP file from LSDyna971 R7.0.0 fails to import in HyperView.
- Component names appear differently upon loading DSY file compared to FEMZIP DSY file.
- Airbag particles blowing up while animating DSY file and FEMZIP DSY file.

13.0 Release Notes - HyperGraph

New Features

- Support for Eleoutdet ASCII file.

Resolved Issues

- Missing chamber definition in the airbags for ABSTAT subcase in the binout reader
- SAE filter with end data reverted behavior to same as HyperWorks 12.0.
- Unable to read rwforc subcase in binout file.
- Support Elout file with new format of scientific numbers without E.
- Missing node requests in BNDOUT ASCII file.

12.0.115 Release Notes

New Features and Enhancements

- Import reader detects unsupported lines in the deck for control cards and gives warning message upon import.
- In the Model Checker, review of Untied nodes in Contact Tied is now available.

Resolved Issues

- Support of new keyword *MAT_COHESIVE_MIXED_MODE_ELASTOPLASTIC_RATE_TITLE
- Support of new keyword *DEFINE_VECTOR_NODES
- Values of SIG22, SIG33 and SIG23 is read irrespective of the value of LOCAL for *Initial_stress_beam
- If *INCLUDE is followed by *INCLUDE_TRANSFORMATION everything works as intended. However, if the order is changed so that *INCLUDE_TRANSFORMATION is followed by *INCLUDE, only the transformed model was imported. This issue has been fixed.
- Proper import of *Mat_Add_Erosion with optional card containing LCFLD now occurs.
- Proper import of Primer cards *MECHANISM_START and *MECHANISM_END and export after *END upon export now occurs.
- In the Model Checker, checks are run twice if the user switches between error and entity view. This has been fixed.
- In select cases when using the Find Attached (Tied) macro, the issue of finding elements till end of the component has been fixed.
- Sometimes the mass value disappeared after quality indexing. This has been fixed.
- Exporting Ls-Dyna Key-File sometimes exports Mass Element Values as 0.0. This has been fixed.
- Incorrect GUI representation when converting PCOMPG to *Part_Composite has been fixed.
- Connectors update was missing in most cases when converting OptiStruct to LS-DYNA conversion Tool. This has been fixed.
- Resolved status update issue between Error view and HM Entity view of Model Checker.

12.0.113 Release Notes

New Features

- A new part replacement feature is available in this release that is faster, more powerful and accurate compared to previous versions of the functionality. This is available for Ls-Dyna and Engineering solutions Ls-Dyna user profiles only.
- Find attached (Tied): A new feature for find attached based on Ls-Dyna contacts. The new find attached will detect physically connected entities or find attached entities that are considered physical connections via their tied contact definitions in Ls-Dyna.

12.0.112 Release Notes

Resolved Issues

- Issue with reading parameterized *ALE_REFERENCE_GEOMETRY_BIRTH_RDT keyword has been fixed
- Support of -2 and -1 options for ELFORM in *SECTION_SOLID
- False error message during import of *CONSTRAINED_EXTRA_NODES_NODE/SET has been removed.

12.0.110 Release Notes

Support of *PARAMETER and *PARAMETER_EXPRESSION

- *PARAMETER and *PARAMETER_EXPRESSION support is limited to Input/output only for the keyword mapped to all the HM entities except elements.
- Parameterization of entity ID's not supported
- Unsupported parameterized entities are replaced with corresponding values during import to avoid loss of data.
- The variables parameterized will be grayed out in the card editor so that it can't be changed.
- In Macros and Browser if any parameterized variable is changed a warning message appears in the message bar and the value will not be changed.

New Keywords Supported

- *DEFINE_FRICTION
- *DEFINE_CURVE_FUNCTION
- *CONTROL_MPP_IO_LSTC_REDUCE
- MAT111 - *MAT_JOHNSON_HOLMQUIST_CONCRETE
- MAT141 - *MAT_RATE_SENSITIVE_POLYMER
- MAT162 - *MAT_COMPOSITE_MSC_OPTION
- MAT168 - *MAT_POLYMER

- MAT244 - *MAT_UHS_STEEL
 - *ALE_FSI_PROJECTION
 - *INITIAL_ALE_MAPPING
 - *CONTROL_MPP_DECOMPOSITION_OUTDECOMP
 - *CONTROL_MPP_DECOMPOSTION_RCBLOG
 - *DATABASE_OPTION_DISBOUT
 - *DEFINE_CONNECTION_PROPERTIES_ADD
-

Update to Supported Keywords Based on LSDYNA 971 R 6.1.0

- *AIRBAG_HYBRID_JETTING
 - *MAT_34 - *MAT_FABRIC
 - *MAT_100 - *MAT_SPOTWELD
 - *MAT_120 - *MAT_GURSON
 - *MAT_120_JC - *MAT_GURSON_JC
 - *MAT_169 - *MAT_ARUP_ADHESIVE
 - *MAT_ADD_EROSION
 - *CONTROL_SHELL
 - *CONTROL_RIGID
 - *CONTACT_ENTITY
 - *CONSTRAINED_RIGID_BODIES
 - *DAMPING_PART_STIFFNESS_SET
 - *DATABASE_BINARY_FSIFOR
 - *DATABASE_OPTIONS
 - *PART_COMPOSITE
 - *PART_COMPOSITE_TSHELL
-

Reorganization

- Keywords *BOUNDARY_PRESCRIBED_MOTION_SET, *BOUNDARY_PRESCRIBED_MOTION_RIGID are moved from load collectors to Loads (constraints, velocity and acceleration).
 - Keywords *ALE_MULTIMATERIAL_GROUP and *SET_MULTI_MATERIAL_GROUP are moved from Interfaces to Sets.
-

Conversion

- LS-DYNA to RADIOSS conversion using HyperCrash feature introduced with extended coverage for conversion.
-

Unsupported Keyword

- Import reader updated to efficiently detect unsupported keywords and move to HMX file.

Import Solver Options

- Option to name *CONSTRAINED_EXTRA_NODES_NODE and *CONSTRAINED_EXTRA_NODES_NODE with PID_NSID

Export Solver Options

- Option to remove ".###" added for duplicate names during import from getting exported to the solver deck.

HyperGraph New Feature

- Support of NODOUTHF and ELOUTHF subcases in binout file

Resolved Issues

HyperMesh

- Support assembly names with "," and maximum of 100 characters during reading solver decks
- Set created from Mass panel is assigned correct card image based on selected entity type
- Reading of keywords *CONSTRAINED_LINEAR_GLOBAL and *CONSTRAINED_LINEAR_LOCAL from solver decks
- Fail to import solver decks with unsupported keywords
- Loss of data during import of following keywords fixed
 - *AIRBAG_INTERACTION
 - *CONTACT_ERODING_SINGLE_SURFACE_ID
 - *INITIAL_GAS_MIX
 - MAT_32 - *MAT_LAMINATED_GLASS
 - MAT_126 - *MAT_MODIFIED_HONEYCOMB
 - MAT_100 - *MAT_SPOTWELD_DAIMLERCHRYSLER
 - MAT_059 (*MAT_COMPOSITE_FAILURE_{OPTION}_MODEL)
- Extra optional card exported for all *CONTACT keywords as blank lines fixed
- Import reader updated to handle both UPPERCASE and LOWERCASE strings in the keywords
- Missing directional node for beam elements while exporting in different includes fixed
- Import reader updated to import multiple *LOAD_BODY keywords in the deck
- Name of set referred in Group HM entity lost during export is fixed
- 3D Visualization for shell elements of card image *ELEMENT_SHELL_COMPOSITE
- Disable transformation defined in the model before exporting an include file from include browser
- Export an include options sets the default path as include file's original directory where it was imported from rather than the start directory

HyperView

- Error in strain contour of solid elements
- Incorrect %thinning reduction value of -1.##IOE+00
- Error in displacement at first time step
- Name of the components in the Component Browser not as per the input deck

HyperGraph

- Fixed difference in request label between windows and Linux for rbdout file
- Ability to read and plot based upon node groups from a LS-Dyna nodfor file
- Fixed crashes in binout and jntforc reader

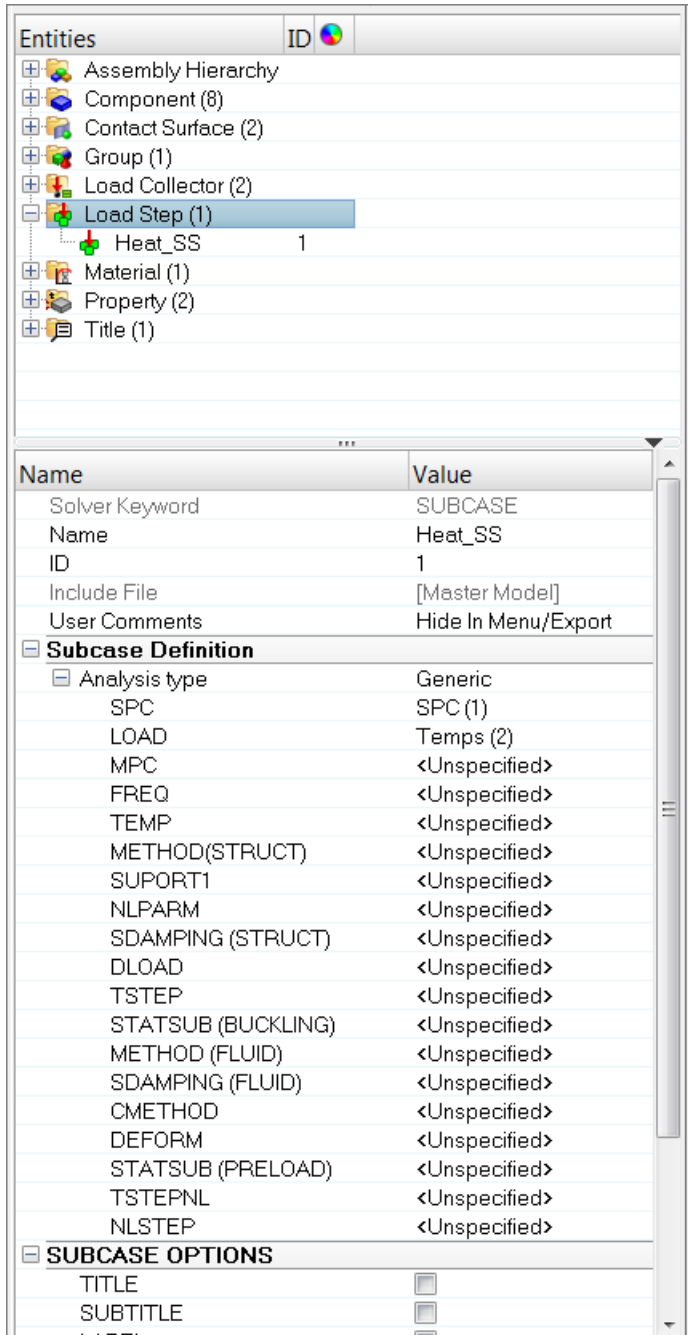
Nastran Interface 13.0 Release Notes

13.0 Release Notes

Entity Editor

Support for Nastran interface in new HyperMesh 13.0 Entity Editor.

Loadsteps displayed in the Entity Editor:



The screenshot shows the Entity Editor interface. The top part is a tree view of entities, and the bottom part is a detailed view of a selected entity.

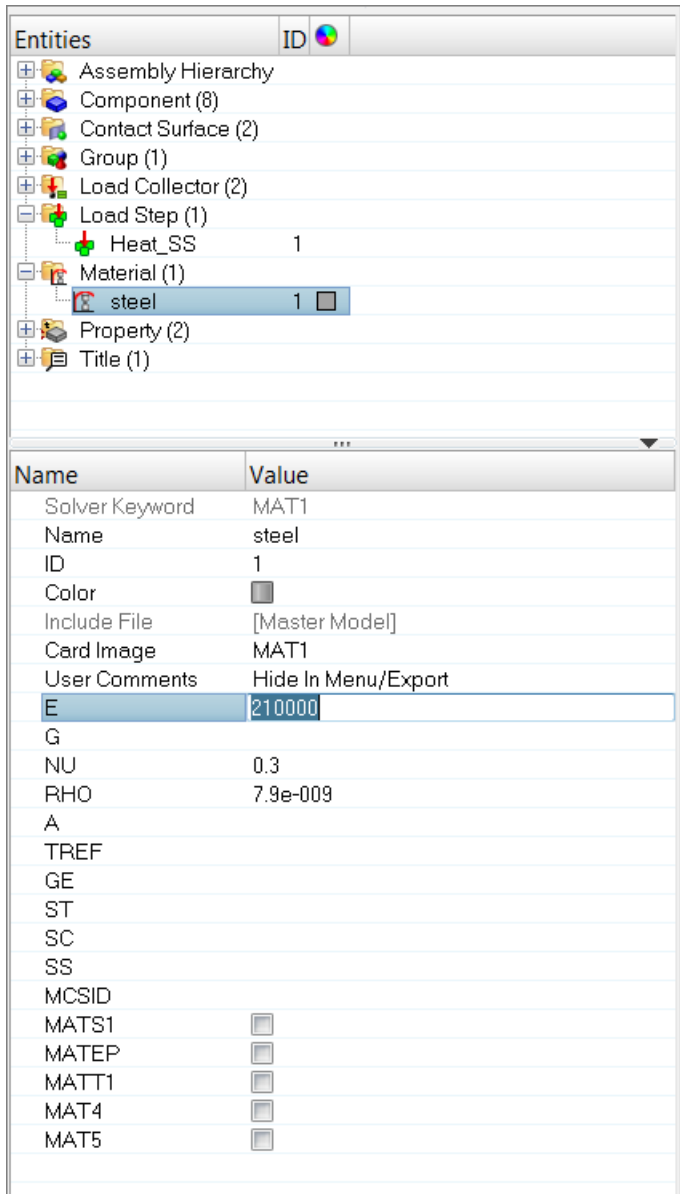
Entities Tree View:

- Assembly Hierarchy
- Component (8)
- Contact Surface (2)
- Group (1)
- Load Collector (2)
- Load Step (1) (Selected)
- Material (1)
- Property (2)
- Title (1)

Entity Details View:

Name	Value
Solver Keyword	SUBCASE
Name	Heat_SS
ID	1
Include File	[Master Model]
User Comments	Hide In Menu/Export
Subcase Definition	
Analysis type	Generic
SPC	SPC (1)
LOAD	Temps (2)
MPC	<Unspecified>
FREQ	<Unspecified>
TEMP	<Unspecified>
METHOD(STRUCT)	<Unspecified>
SUPPORT1	<Unspecified>
NLPARM	<Unspecified>
SDAMPING (STRUCT)	<Unspecified>
DLOAD	<Unspecified>
TSTEP	<Unspecified>
STATSUB (BUCKLING)	<Unspecified>
METHOD (FLUID)	<Unspecified>
SDAMPING (FLUID)	<Unspecified>
CMETHOD	<Unspecified>
DEFORM	<Unspecified>
STATSUB (PRELOAD)	<Unspecified>
TSTEPNL	<Unspecified>
NLSTEP	<Unspecified>
SUBCASE OPTIONS	
TITLE	<input type="checkbox"/>
SUBTITLE	<input type="checkbox"/>

Materials displayed in the Entity Editor:



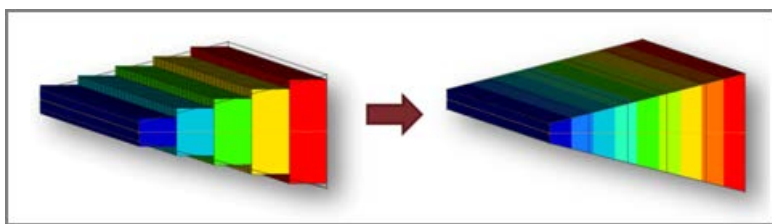
The screenshot shows the Entity Editor interface. The top part is a tree view of entities, and the bottom part is a table of properties for the selected 'steel' material.

Name	Value
Solver Keyword	MAT1
Name	steel
ID	1
Color	<input type="checkbox"/>
Include File	[Master Model]
Card Image	MAT1
User Comments	Hide In Menu/Export
E	210000
G	
NU	0.3
RHO	7.9e-009
A	
TREF	
GE	
ST	
SC	
SS	
MCSID	
MATS1	<input type="checkbox"/>
MATEP	<input type="checkbox"/>
MATT1	<input type="checkbox"/>
MAT4	<input type="checkbox"/>
MAT5	<input type="checkbox"/>

Enhanced Nodal Thickness Visualization

Support for individual nodal thickness 3D visualization and element color mode is now available, making it possible to visualize varying thickness per element instead of constant average value.

- Supports element T_i , ZOFFS values
- Elemental T_i and ZOFFS values supersede property thickness and offset
- Blank elemental T_i inherits property thickness
- Mass calculation uses constant average thickness per elements, as per OptiStruct solver



SUBCOM/SUBSEQ Support for Linear Combination Subcases

- New subcase type *Combination Subcase Delimiter*
- New subcase option SUBSEQ

Bars Panel Enhancements: Offsets in Elemental System, OFFT Flag Support

It is now possible to define offset vectors in elemental system. Additionally, the CBAR and CBEAM OFFT field is now linked to the orientation and offset definitions in the panel.

The screenshot shows the 'Bars Panel' interface. On the left, there are radio buttons for 'bar2' and 'update', and a dropdown menu for 'orientation in displacement' with a red box around it. Below this are input fields for 'x comp = 0.000', 'y comp = 0.000', and 'z comp = 0.000'. In the center, there are input fields for 'pins a = 0', 'property =', 'elem types = CBEAM', and 'offset a:' with sub-fields 'ax = 0.000', 'ay = 0.000', and 'az = 0.000'. On the right, there are input fields for 'pins b = 0', 'offset b:' with sub-fields 'bx = 0.000', 'by = 0.000', and 'bz = 0.000'. A dropdown menu for 'offsets in displacement' is also highlighted with a red box. At the bottom right, there are 'reject', 'review', and 'return' buttons.

The screenshot shows a table of CBEAM elements. The table has columns for EID, PID, GA, GB, X1, X2, X3, and OFFT. The OFFT column is highlighted with a red box. The data in the table is as follows:

EID	PID	GA	GB	X1	X2	X3	OFFT		
3	1	5	61	0.0000000	0.0000000	0.0000000	0.0000000		
				W1A	W2A	W3A	W1B	W2B	W3B
				0.0000000	-1.0000000	0.0000000	0.0000000	-1.0000000	0.0000000

Self-contained Include File Export

New options available in the Model browser's Include View context-sensitive menu (right-click) allows you to export the selected include, or all includes, as self-contained files.

Self-contained include files will contain all nodes and systems that are referenced by entities within the same include, even if such nodes and systems belong to different includes. Because self-contained includes contain all nodes and systems used by entities within it, they can be always imported into HyperMesh on their own without requiring the rest of the model.

New Import Solver Options

- Delete duplicate systems

The screenshot shows the 'Nastran Options' dialog box. It has a title bar with a close button. Below the title bar are three icons: a checkmark, a square, and a circular arrow. The text 'Nastran options:' is followed by several checkboxes and a text input field. The 'Delete duplicate systems' checkbox is highlighted with a red box. The other options are: 'Element to property ratio:' with a text input field containing '3.0', 'Delete duplicate nodes', 'Import comment lines on nodes and elements', and 'Organize unidentified systems into separate collectors'. At the bottom, there are 'OK' and 'Cancel' buttons.

HyperView Reader Enhancements

- OP2, XDB support for Grid Point Element Forces and Moments (12.0.112)

- OP2 CSHEAR results as corner data
 - OP2 CBEAM results as corner data
 - OP2, XDB Support for CTRIAR/CQUADR elements
-

Model Checker

New error check: *RBE3 dof connected to solid*. Flags an error if the DOFs specified on an RBE3 independent grid which belongs to a solid element does not include components 1, 2, or 3, and if it includes components 4, or 5, or 6.

Additional Solver Card Support Updates

- BCBODY: ADVANCE, RIGID, APPROV, GROW continuation lines added
 - MATEP updates:
 - RYIELD field added
 - ANISO, PRESS, VParam, IMPCREEP, CHABOCHE continuation lines added
 - SWLDPRM updates:
 - CHKRUN
 - CNRAGLI
 - CNRAGLO
 - GMCHK
 - GSPROJ
 - GSTOL
 - PROJTOL
 - PRTSW
-

General Changes

- Export performance of single include file has been improved.
 - When importing Nastran Case Control SETs without HM Comments, HyperMesh will no longer create duplicate grid and element set for each one. Instead, HyperMesh will now cross-reference them and assign the appropriate SET type according to the card or cards that reference them. If both types are used, or if HyperMesh cannot identify the type based on references, it will still create both grid and element sets.
 - Hardcoded default values on all cards have been substituted with true solver default designation, represented by a <blank> entry. All previously defined default values will still appear as optional initial value when a card field is first turned on in either the Card Previewer or Entity Editor.
 - HyperView will no longer split linear and non-linear results in two separate subcases when loading OP2 results for non-linear analyses. HyperView will now display all result types in the same original non-linear subcase.
 - Formula to calculate frequency from Complex Eigenvalue analysis in HyperView has been updated from $\sqrt{eigen_real^2 + eigen_imag^2} / 2 * \pi$, to $eigen_imag / 2 * \pi$, which matches solver output. A new OP2/XDB reader option has been added that allows manually reverting to the old formula.
-

Resolved Issues

- Issue with BCSET card referring to SET instead of BCTSET has been corrected.
- HyperView not plotting acceleration results for some Nastran 2012 OP2 files issue has been corrected.

- Failed export of Bulk unsupported cards with very long fields issue has been fixed.
- Issue with Loadstep browser not listing Load Collectors with PLOAD1 loads issue has been fixed.
- ELSTRESS legacy card reading in as unsupported card issue has been fixed. ELSTRESS is now automatically mapped to STRESS.
- TRIM subcase option failing to import issue has been fixed.
- Issue with HyperView not recognizing Nastran input files with large number of comment lines at the beginning of the file has been fixed.
- Error message in Auto Contact tool when selecting components that have squared brackets in name has been fixed.
- NLSTEP subcase data selector not available in non-linear subcase type issue has been fixed.
- Fixed an issue where duplicate loads imported with FE-Overwrite option were not importing correctly for some models.
- HyperMesh can now import nested include files using either a path relative to the file containing its INCLUDE statement, or relative to the master file.
- Fixed an issue where DMIG bulk cards were being imported as CASE_UNsupported cards instead of BULK_UNsupported.
- BOX TRIM resolved issues:
 - Removed **Box Collector** field from Box Trim dialog.
 - Removal of temporary nodes which are outside the bounding box.
 - Closing Box Trim GUI after Trim operation.

12.0.115 Release Notes

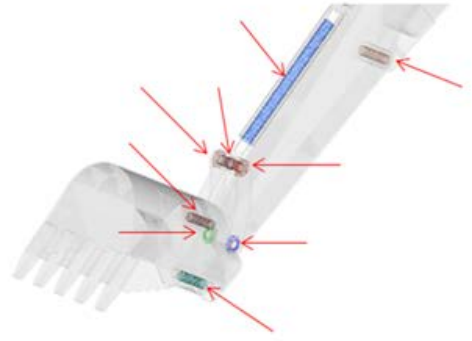
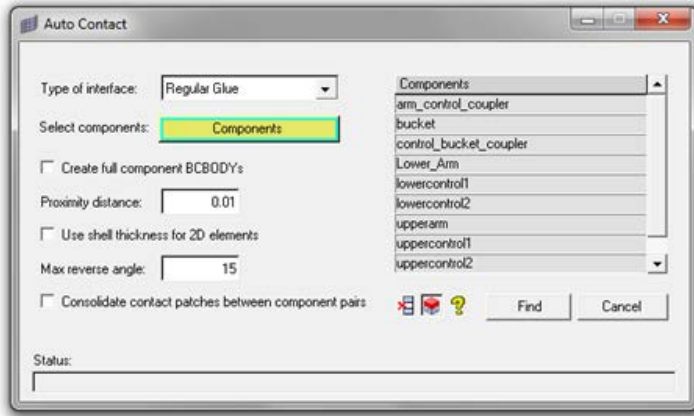
Resolved Issues

- Material MID4 and Z1/Z2 options are now allowed to be defined simultaneously, while ZOFFS and MID4 are now mutually exclusive.
- Issue when importing CBAR elements referencing a system that references another system which is missing from the model has been corrected.
- Issue with HyperView reader crashing when importing a Nastran model containing include files containing only BCONTACT cards has been fixed.

12.0.112 Release Notes

Auto Contact Tool

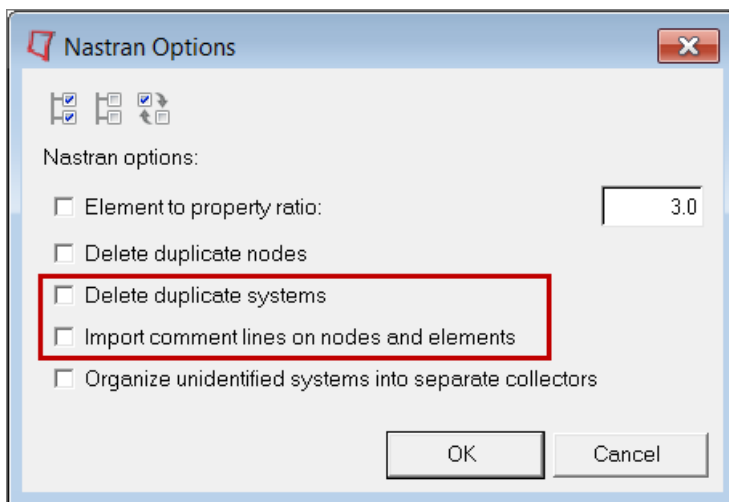
- The Auto Contact tool for Nastran allows you to quickly and easily create one or many contact interfaces at once between several parts of your model. The Auto Contact Browser organizes all interfaces found by the Auto Contact search, where you can review and modify them before accepting any changes.



- This tool is accessible in HyperMesh 12.0.112 via any of the following locations:
 - Tools menu > Auto-Contact
 - BCs menu > Create > Auto-Contact
 - Utility Menu > Nastran1 > Analysis Setup: Auto-Contact
- Nastran Interface: Solver Card Support Updates
 - CHBDYP: New Spring and Mass element types.
 - PHBDY: New Property.
 - CONVM: New extension card inside CHBDYP.
 - PCONVM: New Property.

New Import Solver Options

- Import comment lines on nodes and elements



Model Checker

New error check: *RBE3 dof connected to solid*. Flags an error if the DOFs specified on an RBE3 independent grid which belongs to a solid element does not include components 1, 2, or 3, and if it includes components 4, or 5, or 6.

Additional Solver Card Support Updates

- CHBDYP: New Spring and Mass element types

- PHBDY: New Property
- CONVM: New extension card inside CHBDYP
- PCONVM: New Property

PAM CRASH 2G Interface 13.0 Release Notes

13.0 Release Notes - Pre-processing

Pam-Crash is Now Available in Engineering Solutions

PAM-CRASH is now available as a user profile in Engineering Solutions.

Resolved Issues

- GES browser won't show "review" - nodes on "add entities".
- Names of materials created in HyperMesh is not exported properly.
- The Component table and By Thickness view loses thickness information after element renumbering.

13.0 Release Notes - HyperView

Resolved Issues

- Component names appear differently upon loading DSY file compared to FEMZIP DSY file.
- HyperView crashes reading FEMZIP DSY files that has FPM elements and results.
- HyperView crashes reading FEMZIP DSY files that has more than 1000 steps. New limit on number of steps extended to 10000.

13.0 Release Notes - HyperGraph

New Features

- New THP reader introduced to have improved performance in reading large files.

12.0.115 Release Notes

Resolved Issues

- Issue related to Material ID higher than "7" digits is not supported in HyperMesh for one of the options has been fixed

12.0.110 Release Notes

Supported Keywords Updated in PAM 2010 User Profile

- FRAME
 - MAT 105
 - UNIDIRECTIONAL COMPOSITE
 - FABRIC COMPOSITE B-PHASE PLY MODEL
 - MTJOIN
 - OTMCO
 - BDFORC
 - FPMCONTROL
 - ERFOUTPUT
 - MASS
-

New Features - HyperView Interfacing

- Support of PAM 2012 results in ERHF5 files and DSY files
-

New Features - HyperGraph Interfacing

- Support of PAM 2012 results in ERHF5 files and THP files
-

Resolved Issues

HyperMesh

- Issue with import of ID ranges with extra blanks
- Issue with export of MAT 302

RADIOSS Block Interface 13.0 Release Notes

13.0 Release Notes - HyperMesh

Mid-surface Meshing

- Thickness information from CAD data can be mapped to either the midsurface element or INISHE/INISH3N STRS_F or INISHE/INISH3N STRA_F or INISHE/INISH3N EPSP_F keywords.
- Support for contour and 3D visualization of thickness calculated from the Midsurface Mesh tool.
- Support for offset and scaling capabilities for thickness calculated from the Midsurface Mesh tool.

Entity Editor

- Supported for all keywords mapped in the RADIOSS user profiles except elements.
- Create options available for all keywords mapped to named HyperMesh entities, such as component, property, material, interface, rigidwall, airbag, load collectors, system collectors, vector collectors, sets, block, sensors, and output blocks.
- Edit option available for all keywords, except keywords mapped to elements in HyperMesh.
- Reference entities in keywords
 - New entities that are supported in Entity Editor can be created on fly through a dialog from the current keyword opened in Entity Editor. The new created keyword is referred back in the current keyword.
 - Referenced entities in the keywords can be edited on fly through a dialog from the current keyword opened in Entity Editor.
 - New entity selection browser is introduced to select exiting entities in session with options to filter based on name, ID, card image, include file, and so on.
 - Entity selection browser by default is auto filtered to show entities of valid solver keywords for selection to create error free solver keywords.
 - Show/hide/isolate/Xref functions are available for each referenced entities in the Entity Editor for quick graphics review.

Model Browser

- Solver specific show/hide/isolate/isolate only functionality extended to Model browser like in Solver browser

Encryption Entities

- Support of import/export of encrypted materials, properties and curves
- Support for /KEY keyword

New Supported Keywords

- /MAT/LAW75 based on RADIOSS 12.0.210
- /MAT/LAW77 based on RADIOSS 12.0.210
- /MAT/LAW79 based on RADIOSS 12.0
- /MAT/LAW80 based on RADIOSS 12.0.210

- /LOAD/CENTRI based on RADIOSS 12.0
- /LOAD/PFLUID based on RADIOSS 12.0
- /INIVEL/AXIS based on RADIOSS 12.0
- /PROP/KJOINT2 based on RADIOSS 12.0
- /LEAK/MAT based on RADIOSS 12.0
- /INIBRI/STRA_FGLO and /INIBRI/STRS_FGLO KEYWORD based on RADIOSS 11.0 and 12.0
- /ADMESH/STATE/SHELL based on RADIOSS 11.0 and 12.0
- /ADMESH/STATE/SH3N based on RADIOSS 12.0
- /INISHE/THICK, INISH3/THICK based on RADIOSS 12.0.210
- /INTER/TYPE23 based on RADIOSS 12.0.210
- /OUTP/LSSENSOR based on RADIOSS 12.0.210
- /STATE/LSSENSOR based on RADIOSS 12.0.210
- /ANIM/SHELL/PHI/ALL and ANIM/SHELL/PHI/N based on RADIOSS 12.0
- /ANIM/SHELL/USRi based on RADIOSS 12.0

Updated Supported Keywords

- /MAT/LAW69 based on RADIOSS 12.0.210
- /PROP/TYPE6 based on RADIOSS 12.0.210
- /INTER/TYPE11 based on RADIOSS 12.0.210
- /INTER/TYPE21 based on RADIOSS 12.0.210
- /INTER/TYPE 5 based on RADIOSS 12.0.210
- /INTER/TYPE 7 based on RADIOSS 12.0.210
- /INTER/TYPE19 based on RADIOSS 12.0.210
- /INTER/TYPE12 based on RADIOSS 12.0.210

User Profiles

- Removed the user profiles Fix41, Block42, Block44

Resolved Issues

- SPH meshing tool: Component that has SPH elements has no card image and PROP/SPH is not updated with pitch information.
- /SURF/SEG renumbered during export in Mac.
- TSTOP variable in the /RUN keyword truncated during export.
- Hyperbeam for /PROP/BEAM and /PROP/TRUSS if $I_{xx} \neq I_{yy} + I_{zz}$ is no longer created on import.
- Rigid elements are not updated when the set referred in rigid elements are updated.

13.0 Release Notes - HyperView

New Features/Enhancements

- Support for FEMZIP RADIOSS result files
- Support of element system like RADIOSS solver for shell elements

Resolved Issues

- Crash while exporting deformed shape in RADIOSS format

13.0 Release Notes - HyperGraph

Resolved Issues

- Incorrect data plotted while reading a T01 file that has duplicate labels.
- Loading a T01 fails with: "Error: Unable to recognize: /.../xxxxT01"

12.0.110 Release Notes

New User Profile

- Introduced RADIOSS 120 user profile
- Update the following keywords based on RADIOSS 120 user profile
 - /INTER/TYPE21, /INTER/TYPE7, /INTER/TYPE11, INTER/TYPE19, /INTER/TYPE2
 - /MAT/LAW78
 - /MAT/LAW19
 - /MAT/GAS
 - /MAT/LAW2
 - /MAT/LAW5
 - /PROP/SOLID
 - /PROP/SPR_CRUS
 - /PROP/TYPE17
 - /MONVOL/FVMBAG
 - /MONVOL/FVMBAG1
 - /FAIL/CONNECT
 - /IMPVEL
 - /IMPDISP
 - /SKEW/FIX
 - /SPHGLO
 - /DAMP

New Keyword Support

- INIBRI/EPSP
- INIBRI/AUX
- INIBRI/ENER
- /ANIM/NODA/TEMP
- /ANIM/NODA/DAMA2
- /ANIM/NODA/DENS
- /ANIM/NODA/P

Import

- Provision to import INIBRI cards added to the solver options in the Import Browser.

Reorganization

- Moved all the engine keywords from the engine file to the control cards in HyperMesh. The engine file GUI is removed from the Export and the Solver panel.
- Support of engine file export in batch

Resolved Issues

HyperMesh

- Extra line exported in /INJECT1 keyword.
- /MAT/LAW3, /MAT/LAW4, MAT/LAW49 converted to corresponding material and /EOS/POLYNOMIAL.
- /FAIL/USER1 changes to FAIL/USER2 during import.
- Crash while exporting RADIOSS deck with includes

HyperView

- Error reading gzip result files
- Error in component name while reading result files
- Missing % thinning results when thermal results are in the result file

OptiStruct Interface 13.0 Release Notes

13.0 Release Notes

Entity Editor

Support for OptiStruct interface in new HyperMesh 13.0 Entity Editor.

Groups displayed in the Entity Editor:

The screenshot displays the Entity Editor interface. The top section is a tree view titled "Entities" with a sub-header "ID" and a color selection icon. The tree contains the following items:

- Assembly Hierarchy
- Component (8)
- Contact Surface (2)
- Group (1)
 - top1:base1 (ID: 1, Color: cyan)
- Load Collector (2)
- Load Step (1)
- Material (1)
- Property (2)
- Title (1)

The bottom section is a table showing the properties of the selected entity, "top1:base1".

Name	Value
Solver Keyword	CONTACT
Name	top1:base1
ID	1
Color	
Include File	[Master Model]
Card Image	CONTACT
User Comments	Hide In Menu/Export
Property Option	Property Type
TYPE	SLIDE
SSID	Contactsurf
MSID	top1 (1)
MORIENT	
SRCHDIS	
Adjust Option	String Value
ADJUST	
DISCRET	

Loadsteps displayed in the Entity Editor:

Entities		ID
+	Assembly Hierarchy	
+	Component (8)	
+	Contact Surface (2)	
-	Group (1)	
	top1.base1	1
+	Load Collector (2)	
-	Load Step (1)	
	Heat_SS	1
+	Material (1)	
+	Property (2)	
+	Title (1)	

Name	Value
Solver Keyword	SUBCASE
Name	Heat_SS
ID	1
Include File	[Master Mode]
User Comments	Hide In Menu/Export
Subcase Definition	
Analysis type	Heat transfer (steady state)
SPC	Loadcol
LOAD	Temps (2)
MPC	<Unspecified>
STATSUB(BUCKLING)	<Unspecified>
SUBCASE OPTIONS	
LABEL	<input type="checkbox"/>
SUBTITLE	<input type="checkbox"/>
ANALYSIS	<input checked="" type="checkbox"/>
TYPE	HEAT
EIGVRETRIEVE	<input type="checkbox"/>
EIGVSAVE	<input type="checkbox"/>
EXCLUDE	<input type="checkbox"/>
PADSND	<input type="checkbox"/>
RESVEC	<input type="checkbox"/>
OUTPUT	<input type="checkbox"/>
SUBCASE_UNSUPPORTED	<input type="checkbox"/>

Materials displayed in the Entity Editor:

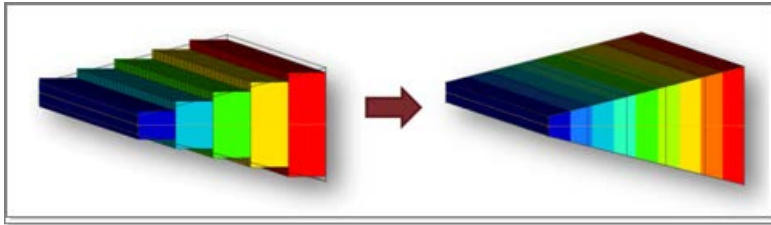
The screenshot shows the Entity Editor interface. The top part is a tree view of entities, and the bottom part is a table of properties for the selected material.

Name	Value
Solver Keyword	MAT1
Name	steel
ID	1
Color	
Include File	[Master Mode]
Card Image	MAT1
User Comments	Hide In Menu/Export
E	210000
G	
NU	0.3
RHO	7.9e-009
A	
TREF	
GE	
ST	
SC	
SS	
MATS1	<input type="checkbox"/>
MATT1	<input type="checkbox"/>
MAT4	<input type="checkbox"/>
MAT5	<input type="checkbox"/>
MATFAT	<input type="checkbox"/>
MATF1	<input type="checkbox"/>
MATX...	<input type="checkbox"/>

Enhanced Nodal Thickness Visualization

Support for individual nodal thickness 3D visualization and element color mode is now available, making it possible to visualize varying thickness per element instead of constant average value.

- Supports element T_i , ZOFFS values
- Elemental T_i and ZOFFS values supersede property thickness and offset
- Blank elemental T_i inherits property thickness
- Mass calculation uses constant average thickness per elements, as per OptiStruct solver



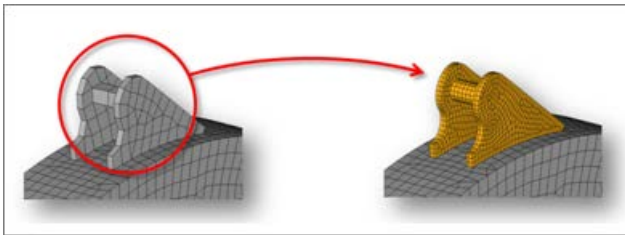
OptiStruct CONTACT Enhancements

- Surface-to-Surface (S2S) contact definition
 - DISCRET: New option inside CONTACT
- Contact thermal conductivity in heat transfer analysis
 - PCONTHT: New extension card inside PCONT

Subcase-specific Modeling and Global-local Analysis (Zooming)

New features in OptiStruct that allow analyzing multiple structures in a single solver run have been added.

- SUBMODEL: New subcase option
- GLOBSUB: New subcase option



SUBCOM/SUBSEQ Support for Linear Combination Subcases

- New subcase type *Combination Subcase Delimiter*
- New subcase option SUBSEQ

Bars Panel Enhancements: Offsets in Elemental System, OFFT Flag Support

It is now possible to define offset vectors in elemental system. Additionally, the CBEAM and CBEAM OFFT field is now linked to the orientation and offset definitions in the panel.

bar2 update
 node A node B

pins a = pins b =

property = elem types = CBEAM

orientation in displacement offsets in displacement

x comp = ax =
 y comp = ay =
 z comp = az =

offset b:
 bx =
 by =
 bz =

↓

	EID	PID	GA	GB	X1	X2	X3	OFFT
CBEAM	3	1	5	61	0.000000	0.000000	0.000000	G00
			W1A	W2A	W3A	W1B	W2B	W3B
			0.000000	-1.000000	0.000000	0.000000	-1.000000	0.000000

Self-contained Include File Export

New options available in the Model browser's Include View context-sensitive menu (right-click) allows you to export the selected include, or all includes, as self-contained files.

Self-contained include files will contain all nodes and systems that are referenced by entities within the same include, even if such nodes and systems belong to different includes. Because self-contained includes contain all nodes and systems used by entities within it, they can be always imported into HyperMesh on their own without requiring the rest of the model.

New Import Solver Options

- Delete duplicate systems.
-

OptiStruct Optimization Enhancements

- Composite manufacturing constraints:
 - Zone-based (GROUP) definition support inside Free Size panel.
 - Thermal/mechanical strain options for composites strain response
 - Mode Shape response
 - **Stress Cluster Constraint** option inside static stress response
 - GPFORCE and SPCFORCE responses
-

Frequency-dependent Materials

Support for OptiStruct frequency-dependent materials:

- MATF1: New extension card inside MAT1
 - MATF2: New extension card inside MAT2
 - MATF8: New extension card inside MAT8
 - MATF9: New extension card inside MAT9
 - MATF10: New extension card inside MAT10
-

Solver Conversion Enhancements

Abaqus > OptiStruct

- Nonlinear springs
- Contact friction
- Surface behavior separation and overclosure
- Orthotropic materials

ANSYS > OptiStruct

- 2D orthotropic materials to MAT8
- 3D orthotropic materials to MAT9ORT

RADIOSS > OptiStruct

- Elements: BEAM, SPRING, TRUSS, BRICK, BRIC20, PENTA4, TETRA4, SH3N, SHELL

- Properties: Beam (TYPE 3/4/8/13), Truss (TYPE 2), Solid (TYPE 14), Shell (Type 1)
- Materials: LAW 0, 1, 2, 36

PERMAS > OptiStruct

- Elements: BECOC, MASS, TRIA, QUAD, SRPING, DAMP, TET, PENTA, HEXE, MPC
- Properties: BEAM, DAMP1/3/6, SHELL, SPRING1/3/6
- Material data: Elastic, Plastic, Hardening, Thermal

Factor of Safety Options for Fatigue Analysis

- FATPARAM:
 - New TYPE=FOS option added
 - New FOS continuation line
- FOS: New global and subcase-specific output card
- MATFAT: New FOS continuation line

HyperView Reader Enhancements

- OP2 CSHEAR results as corner data
- OP2 CBEAM results as corner data

Model Checker

New error check: *RBE3 dof connected to solid*. Flags an error if the DOFs specified on an RBE3 independent grid which belongs to a solid element does not include components 1, 2, or 3, and if it includes components 4, or 5, or 6.

Additional Solver Cards Support Updates

- MATT4: New extension card inside MAT4
- MAT10: ALPHA field added
- MATX28: New extension card inside MAT9ORT
- TICA: New load collector
- CAALOAD: New load collector
- ASSIGN: New H3DCAA option added
- SURF: Option for FACE format added
- CDSMETH: TOL, SSF, RSF fields added. Removed old option SVDP
- MBFRC: Continuation line with G4 option added
- DOPTPRM: APPROX option added
- CONTF: OPTI format added
- CTRIA3/6: TOP/BOTTOM options for ZOFFS added
- CQUAD4/8: TOP/BOTTOM options for ZOFFS added
- PSOLIDX: LAMBDV, MUV, IHKT fields added
- CELAS4: GE, S fields added
- MSGLMT: New MODE option with BRIEF, UNREF, STRICT values
- SWLDPRM updates:

- CNRAGLI
- CNRAGLO
- GMCHK
- GSTOL
- SHOWAUX
- PARAM updates:
 - AGGPCH
 - ASCOUP
 - AUTOMSET
 - BUSHSTIF
 - CMSALOD
 - DISJOINT
 - MASSDMIG
 - ELASSTIF
 - ENFMOTN
 - GRDPNT
 - GRIDFORM
 - MFILTER
 - NLFAT
 - PRINFACC
 - RECOVER
 - RFIOUT
 - SHL2MEM
 - SNAPTHRU
 - SPLREFDB
 - SPLC
 - SPLRHO
 - SPLFAC
 - TRAKMETH
 - TRAKMTX
 - VMOPT

General Changes

- Export performance of single include file has been improved.
- Hardcoded default values on all cards have been substituted with true solver default designation, represented by a <blank> entry. All previously defined default values will still appear as optional initial value when a card field is first turned on in either the Card Previewer or Entity Editor.
- Formula to calculate frequency from Complex Eigenvalue analysis in HyperView has been updated from $\sqrt{eigen_real^2 + eigen_imag^2} / 2 * \pi$, to $eigen_imag / 2 * \pi$, which matches solver output. A new OP2/XDB reader option has been added that allows manually reverting to the old formula.

Resolved Issues

- Abaqus > OptiStruct solver conversion resolved issues:
 - Fixed error message when converting unused composite section properties with orientation/system definitions.
 - Fixed error message when converting models containing names with quotes, semi colons and spaces.
- ANSYS > OptiStruct solver conversion resolved issues:
 - Fixed issue with HyperMesh creating and associating a PMASS property to converted CONM2 elements.
- HyperGraph showing mode numbers instead of frequency values in H3D Eigen Mode plot issue has been corrected.
- Issue with LFREQ/UFREQ fields inside DCONSTR not showing for DRESP2 response has been corrected.
- Mass calculation ignoring for MAT9ORT material issue has been fixed.
- Multiple RANDOM entries are now allowed inside GLOBAL_CASE_CONTROL.
- Issue with HyperMesh panel not reviewing rms-stress optimization response correctly has been fixed.

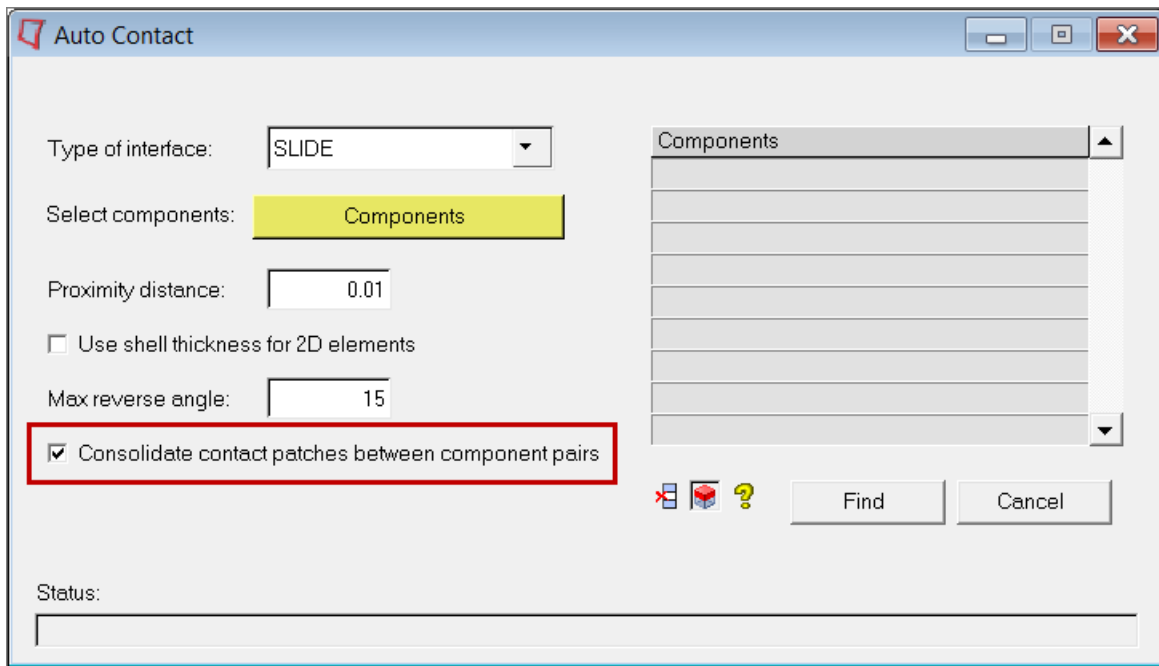
- SET of type elem and subtype prop not importing when referring to PCOMP or PCOMPG issue has been fixed.
- Failed export of Bulk unsupported cards with very long fields issue has been fixed.
- Issue with Loadstep browser not listing load collectors with PLOAD1 loads issue has been fixed.
- CMSMETH SPID_INT field not allowing integer entry of more than 7 characters issue has been corrected.
- Issue with PCONT GPAD option as blank/default being interpreted as THICK has been fixed.
- Fixed an issue with OptiStruct reader ignoring PANEL name in HM Comments during import.
- Import error for some SURF cards generated in SimLab error due to alternate node ordering issue has been corrected.
- Wrong DRESP1 stress code for FRF stress response for shell elements at Z2 location issue has been fixed.
- Multiple entries of PLYDRP continuation line with ALL option are now allowed in DSIZE and DCOMP cards.
- DSIZE import errors due to overflow of PLYDRP PDX, PDY, PDZ values and missing continuation line have been fixed.
- Error message in Auto Contact tool when selecting components that have squared brackets in name has been fixed.
- Issue with Objective panel not allowing subcase selection when using ERP optimization response has been fixed.
- Issue with DVCREL import ignoring EPNAME/FID field with GE value has been fixed.
- Issue with Fluid PFMODE SETDOF selector filtering selection to SET_GRIDC has been fixed.
- Fixed an issue where duplicate loads imported with FE-Overwrite option were not importing correctly for some models.
- HyperMesh can now import nested include files using either a path relative to the file containing its INCLUDE statement, or relative to the Master file.
- Fixed an issue where DMIG bulk cards were being imported as CASE_UNSUPPORTED cards instead of BULK_UNSUPPORTED.
- BOX TRIM resolved issues
 - Removed **Box Collector** field from Box Trim dialog.
 - Removal of temporary nodes which are outside the bounding box.
 - Closing Box Trim dialog after Trim operation.

12.0.115 Release Notes

New OptiStruct Auto Contact Option

There is a new option in the OptiStruct Auto Contact GUI: *Consolidate contact patches between contact pairs*.

This option consolidates any separate contact patch areas that participate in contacts between the same two components, so that the final number of master-slave pairs in the model can be reduced. Activating this option may greatly increase contact search performance for some models.



Resolved Issues

- Material MID4 and Z1/Z2 options can now be defined simultaneously, while ZOFFS and MID4 are now mutually exclusive.
- Fixed an issue with Z0 option inside PCOMP, PCOMPG, PCOMPP not exporting when opening a pre-12.0.110 HyperMesh database file and exporting the model immediately without performing card edit.
- Issue when importing CBAR elements referencing a system that references another system which is missing from the model has been corrected.
- ANSYS > OptiStruct solver conversion resolved issues:
 - Issue with CP_STRUCT entities failing to convert has been fixed.
 - Error message when converting some models with loadstep definitions has been fixed.
 - Fixed issue with converting temperature dependent materials with wrong temperature order, producing solver error
- Fixed issue with converting temperature dependent materials with wrong temperature order from Abaqus to OptiStruct, producing solver error.
- Fixed an issue in the Optimization Response panel where MASS and VOLUME response types were filtering out PSHEAR properties from selection.
- Fixed an issue where Modal Frequency Response subcase created in the Loadstep panel was showing the wrong label in the Loadstep browser.

12.0.113 Release Notes

Solver Card Support Updates

- PCOMP, PCOMPG, PCOMPP: TOP/BOTTOM options for Z0.
- DSIZE, DCOMP: New option for ply drop-off manufacturing constraints (PLYDRP).

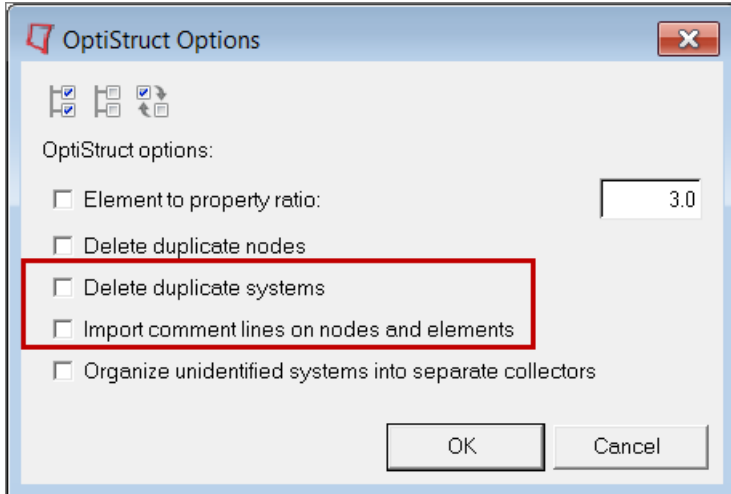
12.0.112 Release Notes

Solver Cards Support Updates

- PCOMP, PCOMPG, PCOMPP: TOP/BOTTOM options for Z0

New Import Solver Options

- Import comment lines on nodes and elements



OptiStruct Optimization Enhancements

- Composite manufacturing constraints:
 - Ply Drop-Off (PLYDRP) support inside free size and composite size panels.

Solver Conversion Enhancements

Abaqus > OptiStruct

- CONN3D elements to CBUSH/PBUSH
- Distributed Coupling surfaces to RBE3
- Offset definitions for composites using SNEG/SPOS keywords
- CLOAD with Real/Imaginary Amplitude

HyperView Reader Enhancements

- OP2 support for Grid Point Element Forces and Moments

Resolved Issues

- An issue related to OSSmooth failing to tetramesh certain parts is corrected
- HyperMesh error message due to renumbering of DTABLE entries has been fixed
- Abaqus > OptiStruct solver conversion resolved issues:
 - Fixed error when converting Abaqus *MPC PIN to OptiStruct RBE2 without DOF values

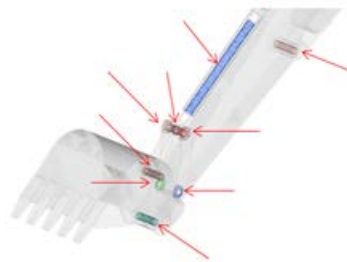
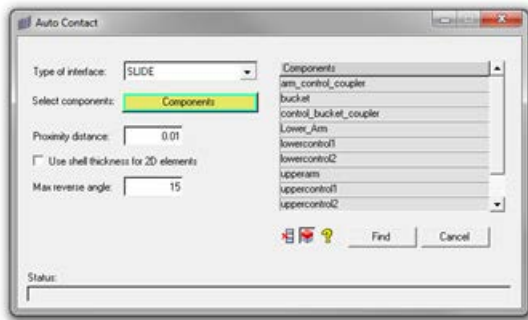
12.0.110 Release Notes

Auto Contact Tool

The Auto Contact tool for OptiStruct allows you to quickly and easily create one or many contact interfaces at once between several parts of your model. The Auto Contact Browser organizes all interfaces found by the Auto Contact search, where you can review and modify them before accepting any changes.

This tool is accessible in HyperMesh 12.0.110 via any of the following locations:

- Tools menu > Auto-Contact
- BCs menu > Create > Auto-Contact
- Utility Menu > FEA > Tools: Auto-Contact

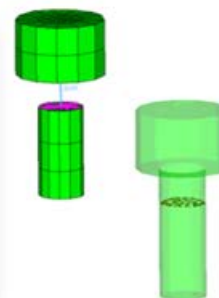
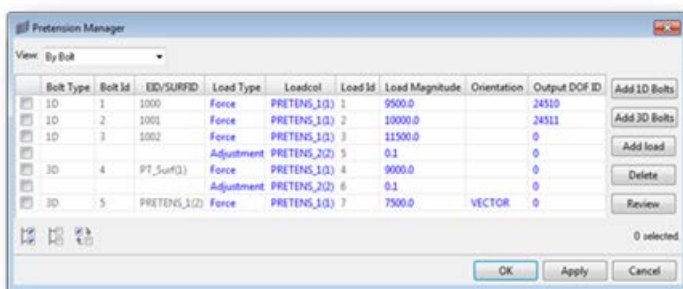


Pretension Manager

The Pretension Manager tool allows you to create and edit 1D and 3D pretension bolt loads and sections for OptiStruct in a quick and streamlined fashion.

This tool is accessible in HyperMesh 12.0.110 via any of the following locations:

- Tools menu > Pretension Manager
- Utility Menu > FEA > Tools: Pretension Manager



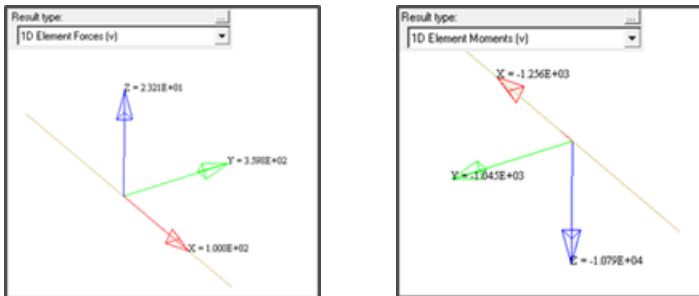
Pretension cards supported for OptiStruct:

- PRETENS: New entity set type
- PTFORCE: New flux load type
- PTADJST: New flux load type
- PTADD: New Load Collector
- PRETENSION: New subcase information entry
- STATSUB(PRETENS): New subcase information entry

1D Element Forces Vector Plots

Vector and tensor plots for 1D and 2D forces and moments, respectively, are now available in HyperView for OptiStruct .op2 and .h3d result file formats.

To visualize the 1D element forces vector plots in HyperView, it is required to load the .fem input deck as model file and .op2 or .h3d as result file, and choose the Advanced Result-Math template.



Model Checker

Several new checks added to the OptiStruct Model Checker, available via Tools > Model Checker > OptiStruct.

New Error checks:

- Property with zero cross section area
- Component with mixed solid and shell elements
- RBE2 with constrained dependent nodes
- Zero-length CBUSH without CID
- MPC with free independent nodes

New Warning checks:

- Material Rho is not defined
- Material Nu is zero
- PCOMP thickness is not defined
- 1D Prop beamsection mismatch
- CONM1 with zero diagonal mass matrix values
- CONM2 with zero mass
- Orientation vector along axis vector

Abaqus to OptiStruct Solver Conversion Enhancements

New enhancements to convert non-linear analysis data, including:

- Gasket elements, properties, materials, loading curves
- Elasto-plastic material curves and properties
- Temperature dependent material data
- Pretension loads and sections

ERP Optimization and Radiated Sound Support

Support for OptiStruct ERP analysis, optimization, and Radiated Sound cards:

- PANELG: New entity set type.

- ERPPNL (12.0): Entity set type.
- FRERP: New response type. Added to optimization responses and Dscreen panels.
- RADSND (bulk data card): New entity set.
- RADSND (subcase information entry): New subcase and Global Case Control entries added.
- SINTENS: New subcase and Global Output Request added.
- SPOWER: New subcase and Global Output Request added.
- SPL: New subcase and Global Output Request added.
- Related PARAMs (12.0):
 - ERPC
 - ERPRLF
 - ERPRHO
 - ERPREFDB
 - RHOCP

Nodal Thickness

3D visualization of elements with nodal thickness is now possible in HyperMesh using the 3D Element Representation visualization mode.

Mass calculation and summary templates now consider element nodal thicknesses as well.

New Import Solver Option: Organize Unidentified Systems Into Separate Collectors

This option affects how systems without HyperMesh comment cards are organized into System Collectors during import.

- When this option is OFF (default), HyperMesh will create a single System Collector containing all systems without HyperMesh comments.
- When this option is ON, HyperMesh will instead create a separate System Collector for each system without HyperMesh comments.



OptiStruct User Profile Fully Replaces RADIOSS (Bulk Data) User Profile

The RADIOSS (Bulk Data) User Profile in HyperMesh has been removed, and is fully replaced by the OptiStruct User Profile. The OptiStruct User Profile can be used to define analysis and optimization solution sequences seamlessly for the OptiStruct solver.

Additional Solver Card Support Updates

- CMSMETH: DMIGDV continuation line added.
- CDSMETH: New Load Collector.
- Output Request updates:
 - MODAL argument added to STRESS, ELFORCE.

- RTYPE argument added to PFMODE.
- PSDF RANDOM option added to ELFORCE, DISPLACEMENT, VELOCITY, ACCELERATION
- SET: SET_RIGID added as new entity set type for creating SET with Type=RIGID.
- HYBDAMP cards:
 - New Load Collector (bulk data card).
 - New Global Case Control entry (I/O option).
- CONTACT updates:
 - New ADJUST option added
 - New Static Friction Coefficient Property Option to define MU1 field.
- PCONT updates:
 - FRICESL option added
 - CLEARANCE option added.
- TIE: New ADJUST option added.
- CONTPRM: FRICESL option added.
- PGAP: FRICESL option added.
- PARAM updates:
 - CONFEL
 - FASTFR
 - INTRFACE
 - EXPERTNL,CNTSTB
- CONTPRM: MTET10 option added.
- GAPPRM: GAPOFFS option added.
- DRESP3: New external response type added to Optimization Responses panel.
- DOBJREF: LOWFQ/HIGHFQ fields added.
- PCNTX5, PCNTX7, PCNTX20: New extension cards added to PCONT.
- CFAST: New mass and rod element types.
- PFAST: New property.
- Loadsteps updates:
 - New Heat Transfer subcase type *Heat Transfer (transient)*. The previously existing heat transfer subcase type is now renamed to *Heat Transfer (steady state)* for consistency.
 - STATSUB(STRUCTURE): New subcase information entry for to heat transfer subcases.

Resolved Issues

- ANSYS > OptiStruct solver conversion resolved issues:
 - Conversion error with BEAM4 containing ADDMAS option has been corrected.
 - PIPE16 now converts to PBEAML with correct dimensions, instead of PBEAM.
 - PCONT converted from CONTA173/174/175 now assigns GPAD=0.0 when KEYOP(11)=0, and GPAD=THICK when KEYOP(11)=1. If KEYOP(12)=5, it converts to CONTACT with Type=FREEZE.
 - CONTA175 conversion is now supported.
 - Conversion of plastic material definitions including MPDATA/TB is now supported.
 - ACEL converting to GRAV without reversing signs issue is resolved.
- CNTNLSUB option missing from Loadstep browser issue has been resolved.
- Model Checker resolved issues:
 - RBE3 with independent node connected only to other RBE3 not being flagged issue has been resolved.
 - RBE2, RBE3 free independent nodes auto correct not deleting free arms issue has been resolved.

- Property is missing material error flagging composite elements even when material is correctly assigned issue has been resolved.
- Improved Moment of Inertia calculation for CONM2 elements with Inertia and Offset values.
- PFMODE not allowing selection of some GRIDC sets issue has been resolved.
- FE-Overwrite enhancements for plies, laminates and tables.
- Updated subcase entry rules and default parameter values in the Loadstep browser.
- PBARL dim values not exporting correctly in long format issue has been resolved.
- Import of certain include files creating additional components named MISC_x issue has been resolved.
- Loadstep panel not allowing you to deselect STATSUB(PRELOAD) from a subcase issue has been corrected.
- HyperGraph not reading .pch files containing ERP results together with other result types issue has been resolved.
- HyperGraph not reading .op2 files without subcase identifier issue has been corrected.

Samcef Interface 13.0 Release Notes

13.0 Release Notes

Resolved Issues

- **Convert Tool:** In 12.110, the RADIOSS Bulk profile loaded after conversion. It has been corrected to load the OptiStruct user profile.
- **Imposed displacement:** When you define a constraint in a loadstep, you have to export an imposed displacement equal to 0 and each component must be exported on different lines. In the previous version this was not the case; this has been corrected in 13.0.

Model Identification Tool 13.0 Release Notes

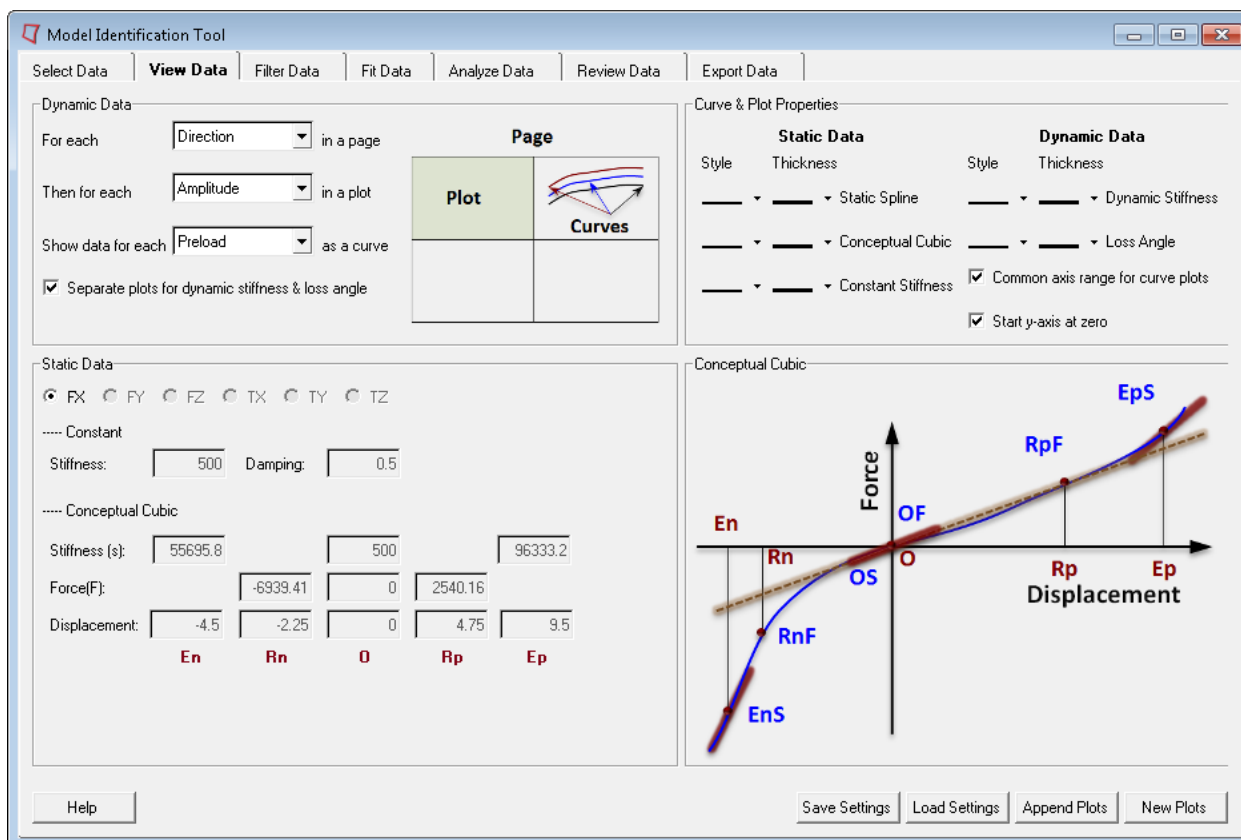
Introduction

HyperWorks HW13.0-HWDesktop contains new features for supporting frequency, and amplitude dependent bushings. The implementation consists of four different components that work together. Collectively, they provide a comprehensive capability for identifying bushings from test data and, subsequently, using bushing models in a simulation. The components are summarized below.

13.0 Release Notes

Model Identification Tool

The MIT manages the workflow associated with fitting experimentally-measured bushing data to analytical bushing models and storing these in ASCII property files. The models in this context are for frequency, amplitude and preload dependent bushings. Two dynamic and three static models are supported. The MIT is launched from HyperGraph.



The MIT is loaded as a preference in HyperGraph. The MIT includes the following seven tabs to support the necessary workflows for fitting bushings:

- **Select Data Tab:** used for specifying experimental data as input.
 - **View Data Tab:** used for examining experimental results.
 - **Filter Data Tab:** used for selecting a subset of the experimental data to be fitted.
 - **Fit Data Tab:** used for fitting data to models for rubber bushing damping or hydromount damping.
 - **Analyze Data Tab:** used for generating the behavior of a set of bushing coefficients.
 - **Review Data Tab:** used for comparing model behavior to experimental data.
 - **Export Data Tab:** used for exporting an analytical bushing model coefficients into an output file.
-

File Input and Output Methods

MIT reads in bushing experimental data that are provided in the form of Tiem-Orbit data files. These are ASCII files that follow the Tiem-Orbit specification.

MIT writes out analytical model behavior in the form of output ASCII files. The most important of these are:

- Output SPD files. Also in Tiem-Orbit format, these contain the results of a virtual test on a virtual bushing. This file is used by the MIT for comparing model performance to measured data.
 - Output GBS file. Also in Tiem-Orbit format, the GBS (General Bushing System) file contains all the data that is required by an MBS solver to simulate a model containing frequency and amplitude dependent bushings.
 - Output LOG file. This is a text file that contains the output of a fitting operation. Details about fitting iterations, optimization iteration history and goodness of fit are included.
-

Operating Modes

- **Interactive mode**

This is the normal mode of operation. A simple user interface guides the user through the steps of inputting test data for fitting, reviewing the test data, fitting the data, reviewing the fit and exporting the fitted bushing parameters to a GBS file. A PDF file containing the fitting summary is automatically exported.

- **Batch mode**

A script, [mit_batch.py](#), is available to run the MIT in a batch mode. Its primary function is to generate one output GBS and/or one SPD file for one bushing. This GBS file can be used for downstream simulation with MotionView/MotionSolve. The two principal modes of operation are:

- Invoke [mit_batch.py](#) with a list of input SPD files (`-i` option).
- Invoke [mit_batch.py](#) with a directory containing a set of SPD files for a bushing (`-d` option).

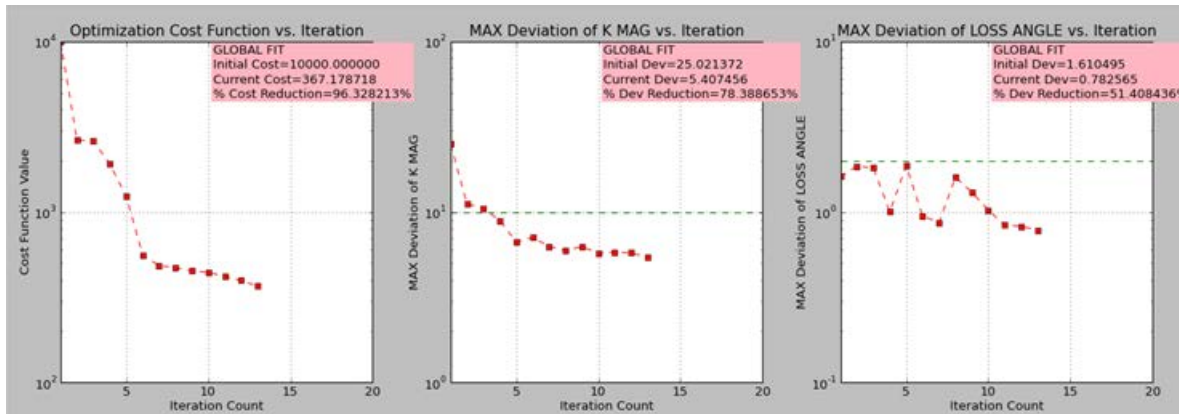
For more information on how to use the batch mode, please refer to the Model Identification Tool User's Guide.

Units Support

Comprehensive Units support including the ability to change input and output Units

MIT Fitting Tool

This tool is invoked by the MIT user interface. It applies sophisticated numerical methods to calculate model coefficients so that deviations between model behavior and known experimental data are minimized. The fitting tool is an extension to HyperGraph.



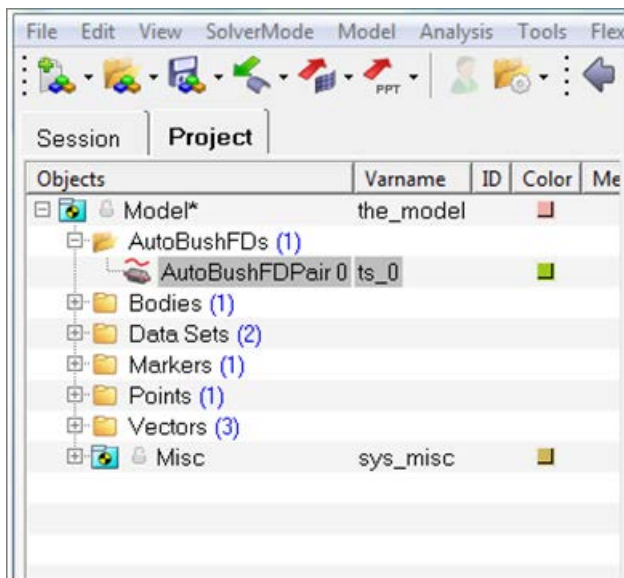
The fitting tool is a utility primarily written in Python that fits experimental data to an analytical model. Some of the features in the Fitting Tool include:

- Reading in bushing input files and generating output files summarizing the fit process.
- Fitting experimental data to a selected model using optimization techniques.
- Generating *virtual experimental* data given a set of bushing model coefficients in a .gbs file.

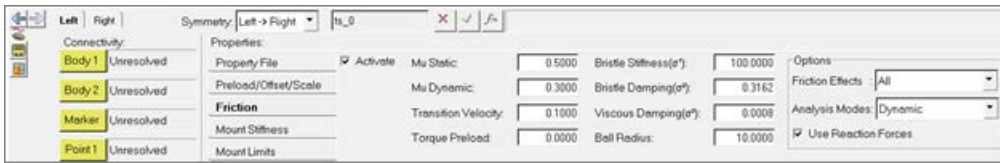
The user does not directly interact with the fitting tool. The MIT provides access to the fitting and analysis functions via the Fit Data Tab and the Analyze Data Tab.

Bushing Model in MotionView

This is a set of panels in MotionView to instantiate and modify frequency, amplitude and preload dependent bushings in a vehicle model.



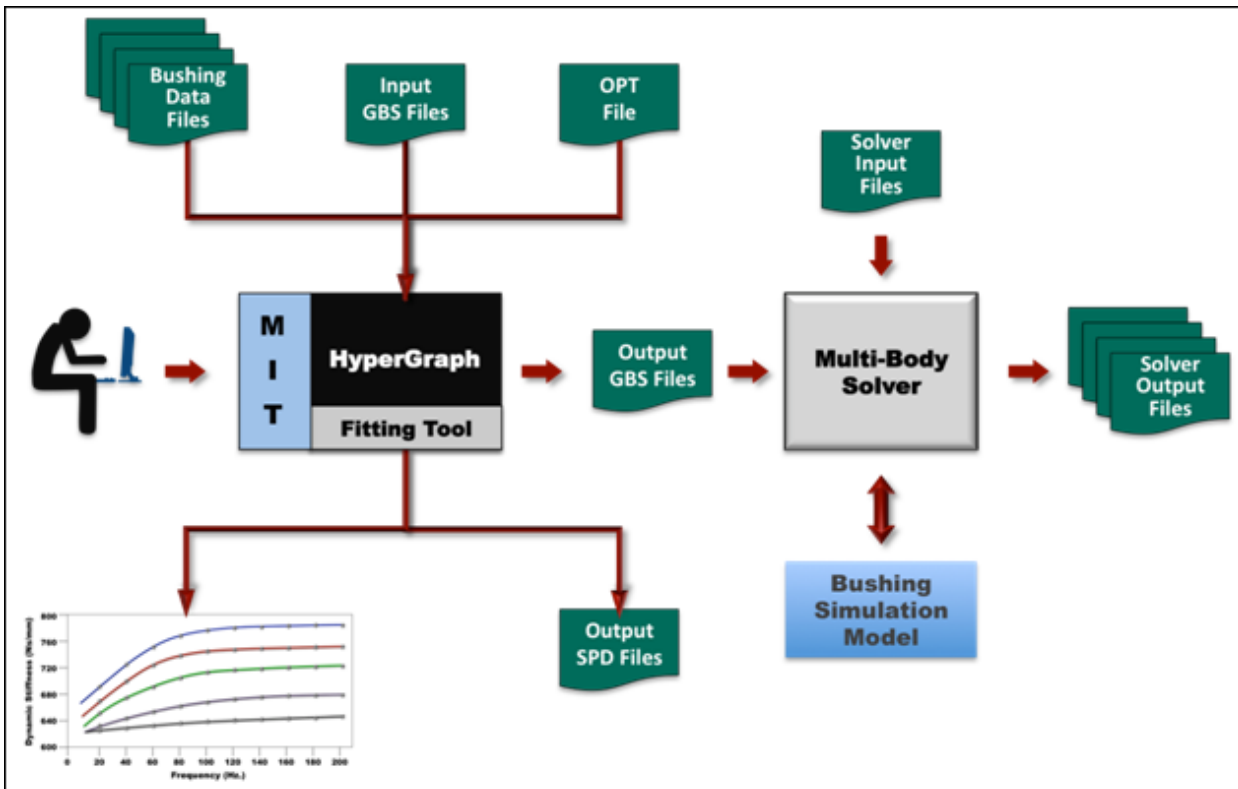
MotionView has been enhanced to contain a new AutoBustFD modeling element. The AutoBustFD is a frequency, amplitude and preload dependent bushing. A set of panels for instantiating an AutoBustFD is now available. When a simulation is to be performed, MotionView exports the AutoBustFD entity to a supported solver such as MotionSolve or ADAMS™.



Altair Bushing Simulation Model

The Altair Bushing Simulation Model is used with fitted bushing models in a multi-body simulation. It is a solver run-time demand load library (.dll or shared library) that works with MotionSolve. Adaptors for Adams/Solver™, MATLAB™ and SIMPACK™ are also available.

The use of this new capability is illustrated in the figure below.



The user interaction with this system is as follows:

- The user interacts with the MIT to fit bushing test data to analytical models. The output of the fitting process is a GBS file that contains the coefficients for the analytical models. The GBS is thus an analytical representation of the test data.
- A full vehicle model is built in MotionView. Bushings are used as flexible connectors at various places in the vehicle.
- The model is passed to MotionSolve, the MBD solver. At run time, MotionSolve loads the Bushing Simulation Model dll (or shared library) to evaluate bushing behavior.
- The output from the solver is a set of files that describe the complete response of the vehicle for the test that was performed.

These are a set of object libraries that simulate bushing behavior in a tool capable of performing multi-body simulations. The Altair Bushing Simulation Model is provided as a neutral .dll that can function with any software that obeys its communication protocol. Extensively tested adaptors are available to make the neutral .dll function seamlessly with these products:

- MotionSolve™
- MATLAB™
- Adams™

- Simpack™

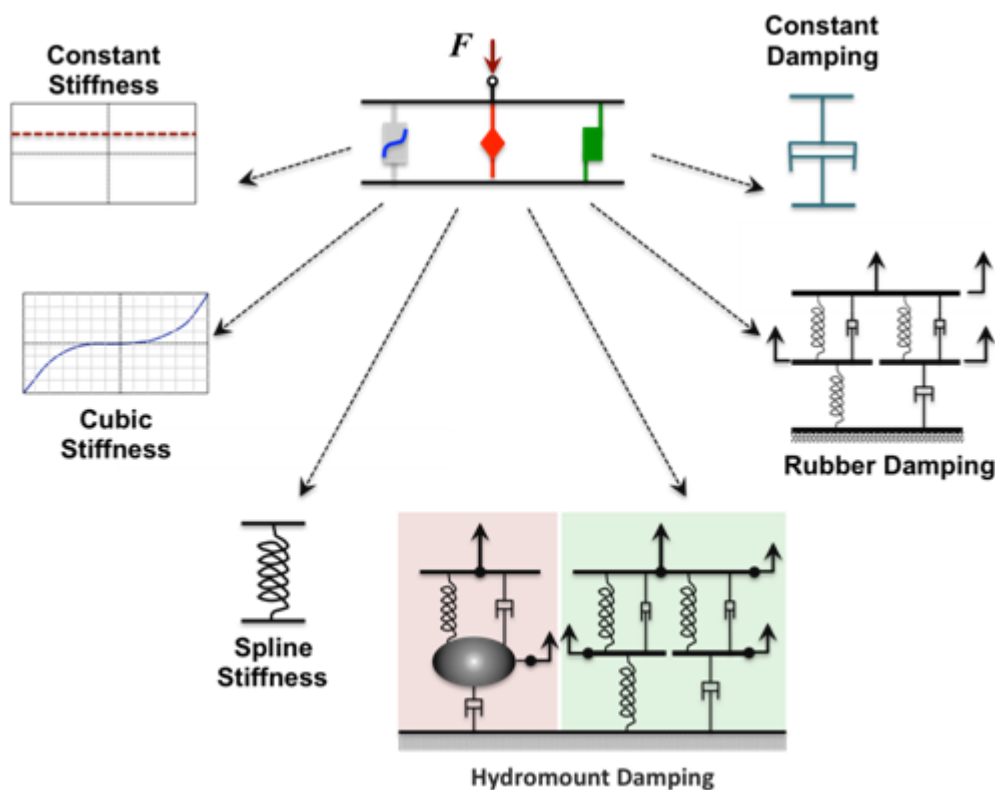
The simulation model includes the following capabilities:

- Three models for static stiffness:
 - Static Spline.
 - Conceptual Cubic.
 - Constant stiffness.
- Three models for dynamic properties:
 - Constant damping
 - Rubber bushing damping
 - Hydromount damping
- A friction model independent of the frequency/amplitude dependence:
 - The LuGre model is used to define the friction parameters.
 - The Dahl model can be obtained simply by setting certain LuGre coefficients to zero.
- A force-coupling model that lets you:
 - Couple bushing forces and torques.
 - Apply a cylindrical or spherical coupling method to the computed forces.
- Local Structural Stiffness:
 - The mount or structure surrounding a bushing may deflect due to the force transmitted by the bushing. Local stiffness models are provided to model this effect.
- Mount Limits:
 - These are bump-stops that model contact in a bushing and are used to limit the range of motion in any direction.
- Displacement Offsets and Scales:
 - Displacement offsets change the origins of the force-deflection curves.
 - Displacement scales expand or contract the force-deflection curves along the deflection axis.
- Preload and Force Scales:
 - Preload offsets the forces or torques at zero displacement.
 - Force scales let you scale the applied force in a direction by a constant.

Supported Bushing Models

The MIT supports three different static models for each direction:

- **Static Spline Model:** This model represents static nonlinear behavior. This data is obtained by reducing the measured static hysteresis shape into a curve. The MIT does not perform this reduction—the user is expected to do this outside of the MIT.
- **Conceptual Cubic Curve Model:** This is a set of two cubic curves that can approximate static nonlinear behavior. The curve coefficients are obtained from the static spline and stored in the GBS file.
- **Constant Stiffness Model:** This is the slope of the static spline at zero deflection.



The MIT supports three different dynamic behavior models for damping: the rubber bushing damping model and the hydromount-damping model:

- **Constant Damping Model:** This model implements a constant valued linear damping function
- **Rubber Bushing Damping Model:** This model is used to describe non-linear hysteretic systems. Patterned after the well-known Bouc-Wen model of hysteresis, this model is able to capture, in analytical form, a range of hysteretic cycle shapes matching the behavior of a wide class of bushings. This model is capable of modeling the dependency of bushing dynamic stiffness and loss angle on the frequency of the input excitation, the amplitude of the input excitation and the operating condition of the bushing.
- **Hydromount-Damping Model:** This is a mathematical model capable of representing realistic hydromount behavior. It consists of a set of non-linear differential equations that describes the rubber and fluid dynamics associated with the movement of the fluid in the decoupler membrane and in the inertia track of the hydromount. Coupling between the rubber and fluid is also modeled.

An SPD (System Performance Data) is provided as input to the MIT. This is a Tiem-Orbit file that contains all the experimental data, including:

- Dynamic Stiffness and Loss Angle as a function of Frequency, Amplitude and Preload.
- Static measurements in the form of a force vs. deflection curve.

The MIT outputs one or more bushing property (.gbs) files through its Export Data capability. You can use these output .gbs files in two ways:

- As an initial guess for design parameters for subsequent fitting.
- As the design parameters for downstream vehicle simulation.

Altair Bushing Simulation Model in Adams/Car™, MATLAB™ and SIMPACK™

Altair offers Adams/Car™, MATLAB™ and SIMPACK™ integration support for simulation models. Please contact your Altair account manager or support@altair.com to learn about this service.

Platform Support

The MIT is available on Windows 7 Professional, Windows 7 Enterprise, and Linux. Both 32- and 64-bit versions of Windows Operating Systems are supported. For Linux, only the 64-bit version is supported. The simulation model is available on the same platforms.

Licensing

The MIT is a utility of HyperGraph and requires that you load HyperGraph prior to launching MIT. The MotionView extensions require MotionView in order to function.

The simulation model is separately licensed and requires six (6) HyperWorks Units in order to function. These Units are leveled against other HyperWorks products that may be running on your machine.

All of these products use the HyperWorks' patented on-demand software licensing system. This centralized licensing model allows multiple users to access HyperWorks products at any time or place. HyperWorks users can run multiple applications for a flat rate, rather than pay for each license in use.

Users Guide and Interactive Help

A comprehensive Users Guide for the MIT is provided with this release. This information is also available through the interactive Help feature in the MIT.

Known Issues

The MIT is a utility of HyperGraph and requires that you load HyperGraph prior to launching MIT. The MotionView extensions require MotionView in order to function.

The simulation model is separately licensed and requires six (6) HyperWorks Units in order to function. These Units are leveled against other HyperWorks products that may be running on your machine.

All of these products use the HyperWorks' patented on-demand software licensing system. This centralized licensing model allows multiple users to access HyperWorks products at any time or place. HyperWorks users can run multiple applications for a flat rate, rather than pay for each license in use.

- **Simulation Model: Static equilibrium for friction model**

Static equilibrium with the friction model is not working properly. It does not work for EFFECT = 'STICTION'.

- **MIT: Displaying results for multiple experiments in the Review tab**

Curves from multiple experiments, when selected, are shown in the same window. This is incorrect. The workaround is to display one experiment at a time.

Virtual Wind Tunnel 13.0 Release Notes

Introduction

Virtual Wind Tunnel is a vertical application to perform wind tunnel simulations of vehicles in an efficient, accurate and robust manner. An intuitive user interface streamlines the process to define parameters for meshing as well as CFD case setup. The definition of physical objects, like rotating wheels or heat exchangers, includes intelligent automation to reduce the user interaction to a minimum.

Automatic report generation is embedded in the system to summarize information about the numerical model, such as mesh statistics or boundary conditions, as well as the aerodynamic results, like lift and drag coefficient.

Based on Altair's CFD solver AcuSolve, Virtual Wind Tunnel delivers highly accurate results and provides a platform for short product development cycles

13.0 Release Notes

New Features

- The parameters of the Run dialog for analysis setup, meshing and results are saved into *.stmod files and can be re-used for further runs.
- Optional extended surface result output for individual parts with user defined output frequency, such as traction vectors for each surface node.
- Parameter to control the mesh transition from boundary layer to the core tetra mesh, which allows a variable height of the last BL layer.
- User defined number of iterations to be considered for computing the average aerodynamic coefficients, such as drag coefficient Cd.
- The Run History table provides an overview of all performed runs and access to the results reports.

Enhancements

- Further improved the robustness of the automated meshing capabilities, especially the mesh intersection step to connect the surface mesh model to the wind tunnel ground before volume meshing.
- The parameters for the wind tunnel ground refinement under car have been moved from the Preference section into the Run dialog.
- The format of the *.par file has been changed to XML format, offering more flexibility for further developments.

Resolved issues

- The individual boundary layer (BL) parameters per part are ignored and the global BL settings are applied for all parts which are selected for boundary layer growth.
- The CI value in the overview table of the report is incorrect; the cross coefficient is used instead of the lift coefficient.
- If shell elements are existing inside a non-fluid domain, such as inside the car cabin, the computation of the aerodynamic coefficients (for example, drag) fails during the report

generation step since no flow result data are existing for those elements (only for the XML format).

- For certain configurations, the user modified axis location for the wheels are saved incorrectly into the *.stmod file.
- For the ground refinement of the wind tunnel under the car, the user defined scaling factor (10% by default) is applied only in x-direction but not in y-direction.

Known Issues

- If the body is not touching any side of the wind tunnel, then no refinement boxes are considered for surface meshing of the wind tunnel. The refinement boxes will only be considered for volume meshing.
- Job submission onto a cluster via the VWT user interface is not supported for this release.

SimLab 13.0 Release Notes

System

- Fix window layout – In earlier versions the toolbars, assembly window, and selection list position changed when the application was minimized and maximized. In this version the locations of these windows do not change unless changed by user.
- (Linux only) The model position is retained while saving SimLab database.
- (Linux only) System is enhanced to remember five recently imported geometry data (*.gda) and database (*.slb) files.

CAD Interface

- CATIA R23 is supported.
- Reading of parameters defined in include files supported in CATIA import.
- UG NX 8.5 is supported.
- (Linux and Windows) STEP to Parasolid conversion added under File | Translate.
- (Linux only) CATIA direct import via Cad Nexus is supported through remote server access.

Import/Export

STL

- (Linux and Windows) Reading performance improved up to 60% for large models.
-

VRML

- (Linux only) Import and export user interface options are supported.
-

Facet Geometry

- (Linux only) Import and export user interface options are supported.
-

Abaqus

- (Linux and Windows) Abaqus cards which are not read/supported by SimLab while import, are added in a separate log file.
- (Linux and Windows) Import of surface traction cards - TRVEC and TRSHR supported.
- (Linux and Windows) Import of thread pretension card *CLEARANCE, TABULAR, BOLT supported.
- (Linux and Windows) Import of step and static card parameters from each load steps is supported.
- (Linux and Windows) Import of pretension force and enforced constraint varying across multiple load steps is supported.
- (Linux and Windows) Import of OP=NEW/MOD parameter from force, pressure and constraints cards supported.

- (Linux and Windows) Import of TYPE=SURFACE TO SURFACE parameter from contact pair card is supported.
- (Linux and Windows) Import of pyramid element - degenerated C3D8 supported.
- (Linux and Windows) Previously while reading DLOAD card (element set format), SimLab will group the element sets if magnitude is same and create single pressure. Now this behavior has been modified to create multiple pressure entities.
- (Linux and Windows) Issues in reading degree of freedom values from kinematic and distributing coupling cards were fixed.

OptiStruct

- (Linux and Windows) OptiStruct import is supported.
- Import of pyramid element (CPYRA) is supported.

PERMAS

- (Linux and Windows) Import enhanced to read MPC, mass, gasket material, surface interaction, spring and load cases.
- Import of pyramid element (PYRA5) is supported.

ANSYS

- (Linux and Windows) CDB format - Multiple EBLOCK reading supported.
- (Linux and Windows) Option to create bodies based on material or element types is added.

Nastran

- Import of pyramid element (CPYRAM) is supported

ADVENTURECluster

- (Linux) Import of nodes, elements and sets are supported.

Graphics/User Interface

- (Linux) File | Settings" dialog is moved to "View | Preference".
- View/Preference
 - Mouse: Additional CATIA mouse settings supported - Included left mouse click for zoom and rotation.
 - (Linux) Meshing: **Use New Mesh Control Dialog** toggle has been added. Disabling this option will display the old Mesh Control dialog.
 - (Linux) Solver: **Create single contact between two bodies** toggle has been added. Enabling this option will merge the contacts between two bodies into single contact.
 - (Linux) Options: **Allow user to enter tolerance in Meshing | Element Intersection** toggle has been added. This option will allow users to enter tolerance value while checking element intersection.
- (Linux) Support enhanced to turn off the filters for better rotation performance.
- (Linux) **Pick through transparency** and **Deselect only** option moved under Picking type.

- (Linux and Windows) Support added to select multiple bodies in assembly tree (Shift selection or hold mouse button and drag).
- (Linux and Windows) Mouse middle button support to change rotation center.
- (Linux) Fillet Loop filter is added to identify connected FEM fillets.
- (Linux and Windows) Rotation performance improved for entities with LBC marker.
- Option added to define cylinder using face group and edge group.
- (Linux and Windows) Option added to remove layer of face/element is added in addition to add layer functionality.
- Option added to calculate radius based on 3 vertices.
- **Pick through transparency** and **Deselect only** option moved under Picking type.
- (Linux) Render Mode | FEM Model | Free Edge Boundary
 - This option is added to display free edges without elements.
- (Linux) Show | Distance
 - Local Coordinate System (LCS) support is added.
- (Linux) Show | Radius
 - Option added to calculate radius based on 3 vertices.
- (Linux) Show | Entities | Elements | Nearby
 - The option is used to identify the nearby elements of the given surface elements. It can also display nearby solid elements by enabling the **Include Solid Elements** option. Identification of elements is based on the distance.
- (Linux) Show | Entities | Bodies | By Elem/Node
 - The option is used to show/hide the bodies by using element/node.
- (Linux) **Cutting Plane** user interface options are enhanced.
- (Linux) Space key support for the options **Edge Merge**, **Edge Split** and **Face Merge**.
- (Linux) Visualization of failed solid elements is improved.

Geometry

Edge | Create | Edge Offset

- (Linux and Windows) Number of layers option added for Tri element type.

Face | Create | Create Fillet

- (Linux and Windows) Support added to select multiple disconnected edges / edge loop as input.

Face | Remove Fillet

- (Linux and Windows) Added a button to flip the identified direction edges.
- (Linux) This option is enhanced to remove multiple disconnected fillets in a single operation.

Face | Show connected

- (Linux and Windows) Tool to identify the connected faces of input surface element or face. Connected faces will be identified up to non manifold region.
-

Body | Create | Sweep

- (Linux) Helical Gear – This option is used to create helical gear of hex/wedge elements.
- (Linux) Helical Spring – Orientation support given for both clockwise and anticlockwise. Previously it was supported only for clockwise direction.

Body | Create | From Solid Elements

- (Linux) Solid elements in a body can be grouped and created as a separate body using this tool.

Body | Create | Ellipsoid

- User interface enhanced.
- Option added to specify the element type for ellipsoid creation.
- Vector components of the ellipsoid are printed in the output window.

Body | Show Features

- (Linux and Windows) Option added to identify open and close partial cylinders.

Body | Break

- (Linux and Windows) Input body name is retained after break operation.

Body | Create | Drill Hole

- (Linux) Option supported to drill hole on the hex body.

Body | Show Duplicate

- Option added to find duplicate bodies based on the bounding box.
- Duplicate body pairs can be easily visualized using output list window.

Model | Sew

- (Linux and Windows) Support extended for meshed sheet bodies.

Mode | Compare

- (Linux and Windows) Enhanced to compare CAD- FEM model.
- (Linux and Windows) For CAD-CAD comparison, option added to check topology in addition to projection.
- (Linux and Windows) Option to consider the mid node position for comparison.
- (Linux and Windows) Fringe plot support added to easily visualize the difference.

Transform

- (Linux and Windows) Basic transformation operations supported for both CAD bodies and coordinates (translate, rotate and mirror).

Align

- Simplified the user interface.
- Option added to translate face from fixed planar face to movable planar face.

Meshing

Mesh Control

- (Linux and Windows) Mesh control user interface has been enhanced.
- (Linux and Windows) Include mesh control is added as a separate mesh control. Previously this was available in body mesh control.
- (Linux and Windows) Preserve mesh control supported for quad4 elements.
- (Linux and Windows) Input body name is retained after breaking using region mesh control.
- (Linux and Windows) Orient element edges along the same direction for faces with fillet and iso line mesh control by using **SL_CONVERT_ISOPATTERN=1** and **STS_WRITE_FILLET_FACES=1**.
- (Linux and Windows) Iso mesh pattern supported for non regular 4 sided faces.

Surface Mesh

- Added the option **Create surface mesh for multiple sheet bodies and merge into a single body** in View | Preference | Meshing to merge multiple sheet bodies into single body after meshing.

Local Remesh

- (Linux and Windows) Performance improved up to 80% while re-meshing topological edges / element input for large models.
- (Linux and Windows) After re-meshing the tri3/tri6 elements, nodes are projected on to the geometry.
- (Linux and Windows) Face and edge mesh control are considered during local re-mesh.

Free Edges

- (Linux) Free edges and non-manifold's information will be displayed in the output window with body names.

Element Intersection

- (Linux and Windows) Option added to get tolerance as input.

Auto Cleanup | Intersection

- (Linux and Windows) Intersection check enhanced to consider shared faces.
- Body pairs identified during intersection can be easily visualized using output list window.

- (Linux) The option is used to resolve the intersections within body and across the bodies.
-

Volume Mesh

- Added an option **Fill hollow space in solids when volume meshing** in View | Preference | Meshing to fill the hollow space inside solids.
-

Qcheck

- (Linux and Windows) Aspect ratio failed TRI elements cleanup using local re-mesh is supported. An **Options** is added to dialog, which allows the user to set the method by which the cleanup can be done.
 - (Linux and Windows) Aspect ratio clean up performance improved.
 - (Linux and Windows) "Height" quality is supported for TET elements.
 - Pyramid element - Volume calculation supported.
 - (Linux) Remembrance support added for the quality parameters and their values.
-

Tools | Axi-Symmetry Meshing

- (Linux) The option is enhanced to the CAD axi- symmetric bodies.
 - (Linux and Windows) Axi symmetry meshing is supported for partially revolved body.
 - (Linux and Windows) Automatic transfer of face color and attributes from CAD to mesh body is supported.
-

Tools | Layered Elements

- **Ignore Neighbor Faces** option added to force the output mesh not to follow the selected faces.
-

Tools | Wrapper Mesh

- **Feature angle** option added for wrapper mesh
 - (Linux and Windows) Added an option to specify the element type (tri3/quad4).
 - (Linux and Windows) Local wrap performance is enhanced.
 - (Linux and Windows) Added an option to preserve internal face during local wrap.
-

Tools | Create Pyramid

- Tool to create pyramid elements by selecting quad4 element surfaces.
-

Tools | M X N Meshing

- (Linux) Mesh size can be given as input. By default number of elements or mesh size is taken from the existing mesh along the direction and it is displayed in the user interface.

Assembly

Join

- (Linux and Windows) **Undo** support added for all join operations.
-

Body union

- (Linux and Windows) User interface is enhanced.
 - (Linux and Windows) Tri6 element input is supported for all Boolean operation.
 - (Linux and Windows) Added an option to remesh the shared edges.
-

FEM

Node | Move

- (Linux and Windows) Added a support to move the nodes based on element normal.
-

Node | Move | To Curve

- (Linux) **To curve** option added to project the nodes or nodes of the source edges on to the target edges.
-

Node | Move | Radial

- (Linux) This option is used to move the nodes in the bodies/faces/edges/nodes along the radial direction.
-

Element | Modify

- (Linux and Windows) Performance improved up to 80% for Tri - Quad and Quad - Tri conversion.
-

Renumber

- Added a support to renumber face based node sets and element sets.
-

Tools | Modify Features | Imprint sphere

- Added a support to specify the sphere depth.
 - Two different patterns are added to imprint the sphere.
-

Tools | Move Face

- Added an option to move the face nodes along normal to the element.

Analysis - Structural and Thermal

Coordinate

- Option to create local coordinates for each node of the selected face is added.
 - The Z axis of the coordinate will be along the node normal.
-

Loads | Force

- Equal distribution type support added. Equal distribution type will equally distribute the load to all the nodes of selected face/edge.
-

Loads | Thermal

- (Linux and Windows) Face input supported.
-

Surface Traction

- (Linux) Tool added to define General surface traction and Shear surface traction for Abaqus.
-

Surface Interaction

- (Linux and Windows) Option to define Friction dependent parameters - Slip Rate, Contact Pressure and Average Temperature is supported.
-

Contact

- (Linux and Windows) Option to trim both master and slave elements is added.
 - Contact dialog options (Trim, Surface Interaction, Press fit and Abaqus parameters) are remembered.
 - (Linux and Windows) Option to specify OptiStruct Adjust parameter added.
 - (Linux and Windows) File based contact supported for ADVENTURECluster Solver.
 - (Linux and Windows) Option to group all the face pairs that are identified while creating contact between two bodies as single contact is added in preferences. This option helps to reduce the objects in the LBC tree.
 - (Linux and Windows) **Convert To OptiStruct** option added in the right-click contact menu to convert all the contact solver parameters to OptiStruct.
 - (Linux) **Set master/slave surface name** option added in the right-click contact menu to define the master and slave surface names.
 - Face/body input supported for PERMAS node-node contact. Master and slave surface are defined as node set and referred in contact.
 - Query contact user interface enhanced.
 - Color based visualization of contact surfaces (master/slave) and contact types is supported. Contact types are supported for Abaqus, OptiStruct and ADVENTURECluster solvers.
 - Elements participating in contact can be visualized.
 - Option to set Master / Slave surface names for Abaqus and PERMAS contact.
-

Material & Property

- (Linux) Option to define multiple gasket unloading curves added.
-

Sets

- Sets user interfaced has been enhanced.
 - Sets specification and database is now supported for region based sets.
-

Export

- Abaqus
 - (Linux and Windows) Pretension force and enforced displacement nodes are grouped into a node set and then referred in the *CLOAD and *BOUNDARY cards. This format will be easy for the user to edit the magnitude values.
 - (Linux and Windows) Added support for exporting HEX20 Gasket element as GK3D18 Abaqus elements.
 - Pyramid elements export supported.
 - (Linux and Windows) Set and surface names starting with numbers will be automatically prefixed with **SL_**. This helps in avoiding errors in the Abaqus solver.
 - (Linux and Windows) Option to control material, property and sets export is added under the Abaqus Export - None option.
- OptiStruct
 - (Linux and Windows) File extension *.fem is supported.
 - Pyramid elements export supported.
 - (Linux and Windows) SOLVTYP parameter supported to select Matrix Solver Type.
 - Option added to specify the RAM allocation while spawning the solver.
- ANSYS
 - (Linux and Windows) CDB format - nodes, elements and sets writing supported.
- Nastran
 - Pyramid elements export supported.
- PERMAS
 - (Linux and Windows) Support added to export the LBC names based on 12/40 char in PERMAS export option.
- Marc
 - Issue in material card format is fixed.

Results

Import Results

- Abaqus 6.13 results import supported.
 - Abaqus normal model results import - all the modes are displayed in results tree view.
 - H3D results reading is supported for OptiStruct results.
-

XY Plot

- Multiple node support is added for load case plot type.

Tools | Bore Distortion

- Distortion Order Summary Plot is added. This is to visualize all the bore distortion orders based on user specified depth.

Tools | Identify Regions

- Results region within the threshold value can also be identified.

Special

Bolt Modeling

- Connect Head and Thread
 - Iso spring connections is supported to connect the Head and Thread RBE
- Thread Pretension
 - Thread pretension parameters are supported in Contact definition file.
- (Linux) Thread bolt
 - **Top and Bottom RBE with bar** special type is supported.
- (Linux) Solid Pretension
 - Option added to specify pretension bolt adjustment length/enforced constraint X displacement value

Excite

- Crank Fillet
 - Two new patterns supported
 - (Linux) Multiple crank fillet patterns are supported
- Crank Bay Model
 - (Linux) Crank Bay Model option is supported.

Power Train

- Membrane Mesh
 - Tool to create layer of membrane elements at the inner bearing surface
- Bearing
 - Tool supported to create main bearing with Hex8/Hex20

Weld

- Tool added to create bead weld between two Quad surfaces.

Trials

Find Crack/Wedge

- (Linux and Windows) The user interface was simplified to identify cracks and wedges separately. Cracks and wedges can also be identified for Quad elements.

Validate Attribute

- (Linux and Windows) Option to validate the circular and cylindrical attribute present in the selected body

Midmeshing

Sketch | Sketch

- Option added to create circle using center node
- Option added to create line using 2-Nodes and 2-Vertices

Automation

New Tools

The following is a list of tools newly supported for automation:

- Import/Export
 - File | Import | ADVC
- Graphics/User Interface
 - Show | Entities | Nodes | By Region
 - Show | Entities | Elements | By Region
- Geometry
 - Body | Edit Parasolid
- Meshing
 - Tools | MxN Meshing
- FEM
 - Tools | Logo Removal
- Results
 - Tools | Strain Gauge
- Trials
 - Find Crack/Wedge
 - Correct Body Face Normal

Enhanced Tools

The following is a list of tools enhanced for automation:

- Import/Export
 - File | Import | CATIA - **Include Files** option supported
- Geometry
 - Transform - Coordinate transformation supported for Translate, Rotate & Mirror
- Meshing
 - Tools | Layered Elements - **Ignore Neighbor Faces** option supported
 - Tools | Wrapper Mesh - **Local wrap** option supported
- FEM
 - Node | Move | Radial - **Move Mid-Nodes with Reference to Corner Nodes** option supported
 - Tools | Modify Features | Imprint Sphere - **Sphere depth and pattern** added

- Tools | Move Face - **Normal to Element** input option added
- Analysis
 - Loads | Thermal - Face support added
 - Sets - Script updated for following set types
 - Node Set - **Include Mid Nodes and Use only Surface Nodes** option added
 - Solid Element - **Include solid element attached to face and Include solid element attached to nodes** option added

Scripting

- **GetFirstElementFromFaceGroup:** To get one reference element (First element) from face group
- **GetDirectionFromFaceGroupElement:** To get direction from one reference element (First element) from face group
- **DefinePlaneFromGroup:** To define plane points from face group (or) edge group
- **OffsetPlane:** To offset the plane to specified offset distance

Evolve v2014 Release Notes

New Features

User Interface/Workflow

- New snap control for bounding box center, edges, edge end points, mid-points, and mid-points for Nurbs curves.
- Added new hotkeys for Snapping. Alt+1,2,3,4 toggles corresponding Grid Snaps, Alt activates Curve & Tangency snaps, Ctrl activates End & Mid point snaps.
- Added a list of all the common hotkeys in the Help pull-down menu.
- New Hide/Unhide feature allows the user to hide/unhide objects independently of objects hidden in construction history.
- Alt+H used to Hide/Unhide Source objects in History
- Added new icon and the "i" key will hide all unselected objects, if no objects are selected it will unhide all hidden objects.
- Added option to turn on or turn off pre-highlighting has been added in the Preferences Panel.
- Curves are displayed as a darker value of the surface color.
- Current model units are now displayed in the application titlebar.
- Icon added in each of the modeling windows for Background images and parameters are controlled in the Modeling Tool Panel.
- Added "F" key shortcut for Fit Whole Model if nothing is selected. If an object is selected this will fit to the selected object.
- Added new icon and the "Q" key will Hide/Unhide all the curves in the scene.

Modeling interaction

- Simplified workflow and editing for the Replication tools.
- Mirror plane is locked to Global axis by default.
- Added G2 continuity support for the Round tool.
- Deformation tools are now Parasolid based for more robust editing.
- Volume and Surface Area tools have been combined and based on Parasolid for increased accuracy.
- Added the Un-combine tool to the Tool Panel - Objects are now uncombined into separate solid bodies (if any exist)
- Added Separate Entities tool to explode entities into individual objects. Added hotkey Cmd+U.
- Improved default settings for the Blend Curve tool and added option for 3rd and 5th degree curves to be used.

Rendering

- Minor bug fixes

Input/Output

- Core Technologies translators updated from 2012SP3 to 2013SP3.
- Added new Parasolid based IGES and STEP translators.

Operating Systems

- Added feature for automatic version update check and notification

Inspire v2014 Release Notes

New Features

Analysis

You can now run linear static and normal modes analysis on your model. Visualize displacement, factor of safety, percent of yield, tension and compression, maximum shear stress, von Mises stress, and major principal stress.

Geometry Simplification

A new family of Simplify / Patch tools have been added that allow you to clean up problem areas in the geometry of a model before running analysis or optimization. The new tools can be used to remove imprints, rounds and fillets, and holes and pockets. You can also plug holes and pockets or create patches and bridges.

Concentrated Mass Parts

The new Concentrated Mass Part tool allows you to create a concentrated mass either on a part or at a point in space.

Connectors

It is now possible to create connectors from concentrated mass parts, forces, torques, supports, and displacement constraints to multiple geometry features.

Sketching

You can now move the sketch plane and make new parts from solids.

Push / Pull Faces Tool

The ruler on the Push / Pull Faces tool has been modified so that you can position a face relative to a precise location on your model.

Solid Editing Tools

The old Boolean Combine, Intersect, and Subtract tools have been simplified and moved to the new Solid Edit icon group, and a new Cut tool for cutting solids has been added.

Optimization

The Optimize icon group has been modified. Click the play button to open the Run Optimization window. Click the bridge icon to view the most recent optimized shape.

Shape Explorer

The Shape Explorer has been streamlined and can now be accessed by clicking the bridge on the Optimize icon group. You can also re-analyze the optimized shape using the Analyze button.

Mass Calculation

The mass of optimized parts can now be viewed in the Model Browser if Mass Calculation has been turned on in the Preferences window.